Gas and hydrodynamics
This document has been prepared by the financial support of European Union. Authors from Riga Technical University, Belarusian State University, Qafqaz University, and National Aviation Academy of Azerbaijan are responsible for the content of this document. This publication reflects the views only of the authors, and it cannot be regarded as the European Union's official position.

The book is developed in a frame of the project “Development of Training Network for Improving Education in Energy Efficiency” acronym: ENERGY, grant Nr. 530379-TEMPUS-1-2012-1-LVTEMPUS-JPCR.

Project was approved by the European Commission in frame of the Program Tempus IV – Fifth call for proposals (Programme guide EACEA/25/2011).

**Sub-programme:** Joint Projects

**Action:** Curricular Reform

**Deliverable:** 2.1 Development and translation of study courses within the frame of direction of gas and hydrodynamics.

---

Project coordinator: **Leonids Ribickis**

Editors: **Roberts Neilands, Leonids Ribickis**

Institution: **Riga Technical University**
Contributors

Adalat Samadov Soltan, vice-rector of Educational affairs, associate professor, Aircrafts and Aviation Engines Department: National Academy of Aviation, Bina settlement, 25th km highway, Baku, AZ 1045, Azerbaijan, tel. +99412 4972829

Andrei Chorny, Head of Turbulence Laboratory, A.V. Luikov Heat and Mass Transfer Institute, National Academy of Sciences of Belarus, Minsk, Belarus

Boriss Gjunsburgs, professor, senior researcher, Water Engineering and Technology Department: Riga Technical University, Civil Engineering Faculty, Institute of Heat, Gas and Water, Azenes Str. 16/20, Riga, LV 1048, Latvia, tel. +371 67089253, e-mail: gjunsburgs@bf.rtu.lv

Gints Jaudzems, assistant, researcher, Water Engineering and Technology Department: Riga Technical University, Civil Engineering Faculty, Institute of Heat, Gas and Water, Azenes Str. 16/20, Riga, LV 1048, Latvia, tel. +371 67089253, e-mail: gints.jaudzems@rtu.lv

Nina Karbalevich, associate professor, Belarusian State University, Physical Faculty, Minsk, Belarus, e-mail: karbalevich@bsu.by

Orkhan Huseyinov, Engineering Faculty, Qafqaz University, Khirdalan, Azerbaijan, tel. +99412 4482862

Parviz Abdullayev Shahmurad, associate professor, Head of Aircrafts and Aviation Engines Department: National Academy of Aviation, Bina settlement, 25th km highway, Baku, AZ 1045, Azerbaijan, tel. +99412 4972824.

Roberts Neilands, associate professor, senior researcher, Water Engineering and Technology Department: Riga Technical University, Civil Engineering Faculty, Institute of Heat, Gas and Water, Azenes Str. 16/20, Riga, LV 1048, Latvia, tel. +371 67089253, e-mail: roberts.neilands@rtu.lv

Romans Neilands, associate professor, senior researcher, Water Engineering and Technology Department: Riga Technical University, Civil Engineering Faculty, Institute of Heat, Gas and Water, Azenes Str. 16/20, Riga, LV 1048, Latvia, tel. +371 67089253, e-mail: romans.neilands@rtu.lv

Rzagulu Agaverdiyev Sultan, lecturer, PhD candidate, Aircrafts and Aviation Engines Department: National Academy of Aviation, Bina settlement, 25th km highway, Baku, AZ 1045, Azerbaijan, tel. +99412 4972838, e-mail: Rzogulu@gmail.com, RAgaverdiyev@swt.az

Yadullah Babayev, associate professor, dean of Engineering Faculty, Qafqaz University, Khirdalan, Azerbaijan, tel. +99412 4482862, e-mail: ybabayev@qu.edu.az

Yulia Zhukova, senior researcher, Turbulence Laboratory, A.V. Luikov Heat and Mass Transfer Institute, National Academy of Sciences of Belarus, Minsk, Belarus, e-mail: julia_zhukova@rambler.ru
Content

1. Properties of fluid ................................................................. 11
   1.1. Fluids ............................................................................. 12
   1.2. Units and dimensions......................................................... 13
   1.3. Density and specific weight ............................................... 14
   1.4. Viscosity .......................................................................... 15
   1.5. Surface tension .................................................................. 16
   1.6. Newtonian and non-Newtonian fluids ................................ 17

2. Fundamentals of fluid kinematics and dynamics .............................. 19
   2.1. Conservation laws for the motion ....................................... 20
         2.1.1. Laws of mass, momentum and energy conservation .... 20
         2.1.2. Basic equations: continuity equation, equations of motion and energy .... 23
   2.2. Generalized Newton law of viscous friction ....................... 24
   2.3. Motion equations for viscous continuum: Navier-Stokes equations .... 26

3. The Bernoulli equations .................................................................. 30
   3.1. Euler’s equation of motion ................................................. 31
   3.2. Bernoulli equation for ideal flow ....................................... 32
   3.3. Bernoulli equation for real flow ....................................... 33
   3.4. Various forms of Bernoulli equations .................................. 35
         3.4.1. Energy form ............................................................. 35
         3.4.2. Head form .............................................................. 35
   3.5. Bernoulli equation for gas .................................................. 36
   3.6. Practical application examples .......................................... 37
         3.6.1. Pitot Tube ............................................................... 37
         3.6.2. Siphon ..................................................................... 38
         3.6.3. Prandtl’s static pressure tube ................................... 39

4. Hydraulic losses in pipe flow .......................................................... 40
   4.1. Shear stress in laminar flow .............................................. 41
   4.2. Shear stress in turbulent flow ............................................ 43
         4.2.1. Velocity distribution in turbulent flow ....................... 44
4.3. Shear stress interaction with hydraulic losses ................................................. 45
4.4. Major hydraulic losses .......................................................................................... 47
  4.4.1. Darcy-Weisbach equation .............................................................................. 49
  4.4.2. Hazen-Williams equation ............................................................................... 50
  4.4.3. Hydraulically equivalent pipes ....................................................................... 52
  4.4.4. Pipe networks .................................................................................................. 53
  4.4.5. Major head loss in gas pipelines ................................................................. 56
    4.4.5.1. Hydraulic calculation of a low-pressure gas pipelines ....................... 57
    4.4.5.2. Hydraulic calculation of gas pipelines at high differential pressures ... 57
    4.4.5.3. Head loss in ventilation ducts ............................................................... 58
  4.4.6. Practical application examples ...................................................................... 58
4.5. Minor hydraulic losses ......................................................................................... 64
  4.5.1. Minor hydraulic losses in sudden expansion (Borda-Carnot equation) ....... 65
  4.5.2. Minor hydraulic losses in sudden contraction ................................................ 67
  4.5.3. Minor hydraulic losses in curve ..................................................................... 69
  4.5.4. Minor hydraulic losses in tee ........................................................................ 70
  4.5.5. Minor hydraulic losses in gradual contraction ............................................... 72
  4.5.6. Minor hydraulic losses in gradual expansion ................................................ 73
  4.5.7. Minor hydraulic losses in valves ................................................................... 74
  4.5.8. Minor hydraulic losses and Reynolds number ............................................. 75
  4.5.9. Practical application example ....................................................................... 76

5. Flow in open channels ............................................................................................. 77
  5.1. Uniform steady flow in open channels. Basic equations .................................. 79
  5.2. Normal depth of the flow in channel ................................................................ 84
  5.3. Specific energy of the cross section. Critical depth ........................................... 84
  5.4. Critical slope ..................................................................................................... 87
  5.5. Velocities restriction at open channel design ..................................................... 88
  5.6. Steady non-uniform gradually varied flow in open channels. Basic differential equation ........................................ 89
    5.6.1. General form of basic differential equation ............................................... 89
    5.6.2. Second form of the basic differential equation ......................................... 91
  5.7. Practical application examples .......................................................................... 93

6. Flow in wastewater treatment units ....................................................................... 94
6.1. Specific aspects of wastewater flows ................................................................. 95
  6.1.1. Flow regimes .................................................................................................. 96
  6.1.2. Flow regimes in wastewater treatment units ................................................... 96
  6.1.3. Practical application example - flow rate estimation ........................................ 97
6.2. Bar racks / Screens .............................................................................................. 98
  6.2.1. Types of screens ............................................................................................ 98
  6.2.2. Hydraulic analysis ......................................................................................... 99
  6.2.3. Design channel depth .................................................................................... 100
  6.2.4. Screenings volume ....................................................................................... 101
  6.2.5. Practical application example - screen design velocity .................................... 101
6.3. Grit removal ......................................................................................................... 102
  6.3.1. Hydraulic analysis ......................................................................................... 103
  6.3.2. Grit settling velocity ...................................................................................... 103
  6.3.3. Required length/diameter .............................................................................. 104
  6.3.4. Grit storage time ......................................................................................... 104
  6.3.5. Practical application example - grit chamber design ...................................... 105
6.4. Clarifiers ............................................................................................................... 106
  6.4.1. Sedimentation basin shapes ......................................................................... 106
  6.4.2. Classes of sedimentation .............................................................................. 106
  6.4.3. Settling basin size determination ................................................................. 109
  6.4.4. Ideal sedimentation basin design practice ..................................................... 110
  6.4.5. Detention Time ............................................................................................. 112
  6.4.6. Surface loading rate ...................................................................................... 113
  6.4.7. Solids loading rate ....................................................................................... 113
  6.4.8. Weir loading rate ......................................................................................... 114
  6.4.9. Surface area .................................................................................................. 114
  6.4.10. Practical application example - sedimentation tank design ......................... 114
6.5. Trickling filters ..................................................................................................... 115
  6.5.1. Hydraulic design equations .......................................................................... 116
    6.5.1.1. NRC Formula: ....................................................................................... 116
    6.5.1.2. Eckenfelder Formula .............................................................................. 116
    6.5.1.3. Hydraulic Loading ................................................................................. 117
    6.5.1.4. Organic Loading .................................................................................... 117
6.5.1.5. Flushing Intensity ................................................................. 117
6.5.1.6. Distributor Rotational Speed ............................................. 117
6.5.1.7. Actual velocity ................................................................... 118
6.5.1.8. Pressure head .................................................................... 118
6.5.2. Practical application example - trickling filter design .......... 119
6.6. Activated sludge ........................................................................ 119
6.6.1. Hydraulic characteristics of reactor tanks ......................... 120
6.6.1.1. Plug flow system ............................................................... 120
6.6.1.2. Completely – mixed system ............................................. 120
6.6.2. Hydraulic phenomena in activated sludge tanks ................ 121
6.6.2.1. Factors affecting the hydraulics of flow ......................... 122
6.6.3. Aeration .............................................................................. 123
6.6.3.1. Air piping ....................................................................... 124
6.6.4. Design aspects ..................................................................... 124
6.6.4.1. Required aeration volume ............................................ 125
6.6.4.2. Return activated sludge (RAS) rate ................................. 125
6.6.4.3. Solids retention time (SRT) ........................................... 125
6.6.4.4. Food to mass (F/M) ratio ............................................... 125
6.6.4.5. Sludge wasting rate, mass based .................................... 125
6.6.4.6. Oxygen requirements .................................................... 126
6.7. Sludge Thickeners .................................................................... 126
6.7.1. Hydraulic loading rate ......................................................... 127
6.7.1. Sludge detention time .......................................................... 127
6.7.2. Solids loading rate ............................................................... 127
6.8. Wastewater treatment plant hydraulic profiles ...................... 128

7. Gas and liquid streams diagnosis ................................................. 129
7.1. Flow sensors ........................................................................... 130
7.1.1. Thermal flow meters ............................................................ 130
7.1.2. Hydrodynamic (aerodynamic) flow meters ......................... 136
7.1.3. Electromagnetic flow meters .............................................. 138
7.1.4. Ultrasonic flow sensors ....................................................... 140
7.2. The viscosity of liquids measurement .......................................... 145
7.2.1. Capillary viscosimeters ....................................................... 148
7.2.2. Rotational viscosimeters .......................................................... 152
7.2.3. The falling ball method .......................................................... 154

8. Introduction to the heat conduction theory ........................................ 156
   8.1. The differential equation of heat conduction ................................ 158
      8.1.1. Boundary conditions ......................................................... 158
   8.2. Differential equations of convective heat transfer ......................... 167
      8.2.1. The energy equation .............................................................. 168
      8.2.2. The equations of motion ...................................................... 171
      8.2.3. Continuity equation ............................................................. 175
      8.2.4. Conditions of uniqueness ................................................... 177
      8.2.5. The hydrodynamic and thermal boundary layers ................. 178
         8.2.5.1. Terms of «sticking» ....................................................... 178
         8.2.5.2. The equation of heat transfer ...................................... 178
         8.2.5.3. Hydrodynamic boundary layer ..................................... 179
         8.2.5.4. Thermal boundary layer ............................................... 180
         8.2.5.5. Similarity of the convective heat transfer processes ..... 181

9. Experimental methods for determining of thermo physical characteristics ...... 190
   9.1. Methods for measuring the temperature ........................................ 191
      9.1.1. The study of resistance thermometers and their graduation .......... 192
      9.1.2. Thermoelectric phenomena. Temperature measurement using thermocouples 195
      9.2.1. Non-stationary methods of determining of thermophysical characteristics in initial stage of heat transfer in the absence of heat sources .............. 200
      9.2.2. Determination of the thermal diffusivity coefficient in the regular mode ...... 201
      9.2.3. Determination of thermal conductivity coefficient by a comparative method in the steady state ................................................................. 202
      9.2.4. Determination of thermophysical characteristics in a medium of constant temperature in the presence of heat source .................................. 204
      9.2.5. Non-stationary methods of determining the thermophysical characteristics in the initial stages of heat exchange ........................................ 205
      9.2.6. Determination of thermophysical characteristics of solid materials in regular mode ................................................................. 208
      9.2.7. Determination of thermal conductivity coefficient in the steady state .............. 210
      9.2.8. Pulse methods for determining the thermophysical characteristics .......... 210
12.2. Cooling via internal convective flows................................................................. 274
12.3. Fundamentals of film cooling performance .................................................. 295
  12.3.1. Correlations of film cooling performance............................................... 303
12.4. A conjugate heat transfer method for turbine blade cooling .................. 310

References ..................................................................................................................... 317
1. Properties of fluid
1.1. Fluids

All matter consists of only two states, fluid and solid. By definition, a solid material is rigid. A rigid material tends to shatter when subjected to very large stresses, it can withstand a moderate shear stress for an indefinite period. When a shear stress is first applied to a rigid material, it deforms slightly. But then springs back to its original shape when the stress is unloaded.

A fluid material possesses no rigidity at all and a fluid is a substance whose molecules move freely past each other. More specifically, a fluid is a substance that will continuously deform - that is, flow under the action of shear stress. A small fluid element is unable to withstand and tendency of an applied shear stress to change its shape. A solid will deform under the action of shear stress but it will not flow like a fluid.

Fluids are conventionally classified as either liquids or gases. Liquids and gases differ because of forces between molecules. Liquids take the shape of container and create a surface under the influence of gravity. Gases expand to fill a closed container. The behavior of the liquid is produced by strong attractive force between the molecules. This strong attractive force also explains why the density of a liquid is much higher than the density of a gas. A gas is a phase of material in which molecules are widely spaced. Molecules move freely and forces between molecules are weak. Whereas, a liquid is a phase of material in which molecules are closely spaced and forces between molecules are strong. In Table 1.1 is shown differences between a gas and a liquid.

Table 1.1. Differences between gases and liquids

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Liquid</th>
<th>Gas</th>
</tr>
</thead>
<tbody>
<tr>
<td>Macroscopic description</td>
<td>Liquids take the shape of container and stay in open container</td>
<td>Gases expand to fill a closed container</td>
</tr>
<tr>
<td>Mobility of molecules</td>
<td>Liquids typically flow easily even though there are strong intermolecular forces between molecules</td>
<td>Molecules move around freely with little interaction except during collisions; this is why gases expand to fill their container</td>
</tr>
<tr>
<td>Typical density</td>
<td>Medium; density of water is 1000 kg/m$^3$</td>
<td>Small; density of air at sea level is 1.2 kg/m$^3$</td>
</tr>
<tr>
<td>Molecular spacing</td>
<td>Small-molecules are held close together by intermolecular forces</td>
<td>Large- on average, molecules are far apart</td>
</tr>
<tr>
<td>Effect of shear stress</td>
<td>Produces flow</td>
<td>Produces flow</td>
</tr>
<tr>
<td>Effect of normal stress</td>
<td>Produces deformation associated with volume change</td>
<td>Produces deformation associated with volume change</td>
</tr>
<tr>
<td>Viscosity</td>
<td>High; decreases as temperature increases</td>
<td>Low; increases as temperature increases</td>
</tr>
<tr>
<td>Compressibility</td>
<td>Difficult to compress; bulk modules of liquid water is 2.2x10$^9$ Pa</td>
<td>Easy to compress; bulk modules of a gas at room conditions is about 1.0x10$^5$ Pa</td>
</tr>
</tbody>
</table>
1.2. **Units and dimensions**

We will be describing fluid characteristics that involves quantitatively and qualitatively. A dimension is a category that represents a physical quantity such as mass, length, time, momentum, force, acceleration, and energy. Thus length is a dimension associated with such variables as distance, displacement, width, deflection, and height. There are length, mass, time, temperature, electric current, amount of light, and amount of matter that these quantities are called primary dimensions. Secondary dimensions such as momentum and energy can be related to primary dimensions by using equations. For example, the secondary dimension "force" is expressed in primary dimensions by using Newton's second law of motion, \( F = ma \). The primary dimensions of acceleration are \( L/T^2 \), so

\[
F = ma = M \frac{L}{T^2}
\]  
(1.2.1)

A unit is a particular way of attaching a number of the quantitative dimensions. For example, measurement of volume is a dimension that can be expressed using units of liters. Similarly, measurement of energy is a dimension that can be expressed using units of joules. Most dimensions have multiple units that are used for measurement. For example, the dimension of "force" can be expressed using units of Newtons, pounds-force, or dynes.

There are several systems of units in use and we shall consider two systems that are commonly used in engineering. In the British Gravitational (BG) System the unit of length is the foot (ft), the unit of time is the second (s), the unit of force is the pound (lb), and the unit of temperature is the degree Fahrenheit (°F). The unit of mass is called slug, is defined Newton's second law (force = mass \( \times \) acceleration) as:

\[
1 \text{lb} = 1 \text{slug} \times 1 \text{ft} \times \text{s}^2
\]  
(1.2.2)

The International Systems (SI) of Units is based on the meter (the unit of length), kilogram (the unit of mass), second (the unit of time), and Kelvin (the unit of temperature). The Kelvin temperature scale is an absolute scale and is related to the Celsius (centigrade) scale (°C) through the relationship:

\[
K = C + 273.15
\]  
(1.2.3)

The force unit is called Newton (N), id defined Newton's second law as:

\[
1 \text{N} = 1 \text{kg} \times (1\text{m/s}^2)
\]  
(1.2.4)

Some of the common variables dimensions and their SI units are listed Table 1.2.

Table 1.2. Common dimensions (Crowe et al., 2009)

<table>
<thead>
<tr>
<th>Dimension</th>
<th>Symbol</th>
<th>Unit (SI)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Length</td>
<td>L</td>
<td>Meter (m)</td>
</tr>
<tr>
<td>Mass</td>
<td>M</td>
<td>Kilogram (kg)</td>
</tr>
<tr>
<td>Time</td>
<td>T</td>
<td>Second (s)</td>
</tr>
<tr>
<td>Temperature</td>
<td>T</td>
<td>Kelvin (K)</td>
</tr>
<tr>
<td>Electric current</td>
<td>I</td>
<td>Ampere (A)</td>
</tr>
<tr>
<td>Amount of light</td>
<td>C</td>
<td>Candela (cd)</td>
</tr>
<tr>
<td>Amount of matter</td>
<td>N</td>
<td>Mole (mol)</td>
</tr>
</tbody>
</table>
Whereas some useful conversion factors between different units systems are listed in Table 1.3. Prefixes are used to indicate multiples of the units to avoid very large or very small numerical values. Some of the common prefixes are listed Table 1.4.

Table 1.3. Conversation factors (Kundu et al., 2012)

<table>
<thead>
<tr>
<th>Dimension</th>
<th>Conversation factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>Length</td>
<td>1 m=3.2808 ft</td>
</tr>
<tr>
<td></td>
<td>1 in.=2.540 cm</td>
</tr>
<tr>
<td></td>
<td>1 mile=1.609 km</td>
</tr>
<tr>
<td></td>
<td>1 nautical mile=1.852 km</td>
</tr>
<tr>
<td>Mass</td>
<td>1 kg=2.2055 lbs=0.06854 slug=1000gr</td>
</tr>
<tr>
<td></td>
<td>1 metric ton=1000 kg</td>
</tr>
<tr>
<td>Time</td>
<td>1 h=3600 s</td>
</tr>
<tr>
<td></td>
<td>1 day=86400 s</td>
</tr>
<tr>
<td>Density</td>
<td>1 kg/m³=1.941x10⁻³ slugs ft³=0.06244 lbs/ft³</td>
</tr>
<tr>
<td>Velocity</td>
<td>1 knot=0.5144 m/s</td>
</tr>
<tr>
<td>Force</td>
<td>1 N=10⁶ dyn=0.2248 lbs</td>
</tr>
<tr>
<td>Pressure</td>
<td>1 dyn/cm²=0.1 N/m²=0.1 Pa</td>
</tr>
<tr>
<td></td>
<td>1 bar=10⁵ Pa</td>
</tr>
<tr>
<td>Energy</td>
<td>1 J=10⁷ erg=0.2389 cal</td>
</tr>
<tr>
<td></td>
<td>1 cal= 4.185 J</td>
</tr>
</tbody>
</table>

Table 1.4. Common prefixes (Kundu et al., 2012)

<table>
<thead>
<tr>
<th>Prefixes</th>
<th>Symbol</th>
<th>Multiple</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mega</td>
<td>M</td>
<td>10⁶</td>
</tr>
<tr>
<td>Kilo</td>
<td>K</td>
<td>10⁴</td>
</tr>
<tr>
<td>Deci</td>
<td>D</td>
<td>10⁻¹</td>
</tr>
<tr>
<td>Centi</td>
<td>C</td>
<td>10⁻²</td>
</tr>
<tr>
<td>Milli</td>
<td>M</td>
<td>10⁻³</td>
</tr>
<tr>
<td>Micro</td>
<td>µ</td>
<td>10⁻⁶</td>
</tr>
<tr>
<td>Nano</td>
<td>N</td>
<td>10⁻⁹</td>
</tr>
</tbody>
</table>

1.3. Density and specific weight

The density of a fluid is defined as its mass per unit volume and it is designated by the Greek symbol \( \rho \). This study density is used to characterize the mass of fluids. In the SI unit system, density unit is kg m⁻³.

The value of density can vary widely between different fluids, but for liquids, variations in pressure and temperature generally have only small effect on the value of \( \rho \).

The specific weight of a fluid is defined as its weight per unit volume and it is designated by the Greek symbol \( \gamma \). Water at 20°C has a specific weight of 9790 N m⁻³ and the specific weight of air at 20°C and standard atmospheric pressure is 11.8 N m⁻³. There is the following relationship between specific weight and density:

\[
\gamma = \rho g
\]
1.4. Viscosity

All fluids offer resistance to any force tending to cause one layer to move over another. The viscosity is the fluid property that is a measure of resistance to flow and it is designated by the Greek symbol $\mu$. If we tip over a glass of water on the table, the water spill out before we can stop it. If we tip over a jar of honey, we probably can set it up right again before much honey flows out. This is possible, because the honey has much greater resistance to flow, more viscosity than water.

Consider the motion of fluid is illustrated in Figure 1.4.1 and a fluid element sheared in one plane by a single shear stress $\tau$. All particles are moving in the same direction, but different layers of the fluid move with different velocities. Thus one layer moves relative to another. We assume for the moment that the parallel movements of the layers are in straight line. A particular small portion of the fluid will be deformed from its original rectangular shape PQRS to P'Q'R'S' as it moves along.

![Figure 1.4.1. The motion of fluids](image_url)

However, it is not the displacement of P'Q' relative to S'R' that is important, so much as the angle $\alpha$. The linear displacement is a matter of the difference of velocity between the two planes PQ and SR but the angular displacement depends also on the distance between the planes. Thus the important factor is viscosity gradient, that is, the rate at which the velocity changes with the distance across the flow. Within a flowing fluid, the velocity $u$ of the fluid varies with distance $y$ measured from some fixed reference plan, in such a manner as in Figure 1.4.2.

![Figure 1.4.2. Velocity profile](image_url)
The velocity gradient is given by $du/dy$, where $u$ is fluid velocity and $y$ is the distance measured from the wall. The velocity distribution shown is characteristic of flow next to a stationary solid boundary, such as fluid flowing through a pipe. For the straight and parallel motion of a given fluid, the tangential stress between two adjoining layers is proportional to the velocity gradient in a direction perpendicular to the layers. That is

$$\tau = F/A du/dy$$

or

$$\tau = \mu \frac{du}{dy}$$  \hspace{1cm} (1.4.1)

where $\mu$ is a constant for a particular fluid at a particular temperature. Therefore $\mu$ is related to the shear stress and velocity gradient:

$$\mu = \frac{\tau}{du/dy} = \frac{N \cdot m^2}{(m \cdot s) \cdot m} = N \cdot \frac{s}{m^2}$$  \hspace{1cm} (1.4.2)

Viscosity varies widely with temperature. However, temperature variation has an opposite effect on the viscosities of fluids. The difference is due to their fundamentally different mechanism creating viscosity characteristics. In gases, molecules are sparse and cohesion is negligible. But in the liquids, molecules are more compact and cohesion is more dominate. Thus, in gases, the exchange of momentum between layers brought as a result of molecular movement normal to the general direction of flow, and it resists the flow. This molecular activity is known to increase with temperature, thus, the viscosity of gases will increase with temperature. This reasoning is a result of the considerations of the kinetic theory. This theory indicates that gas viscosities vary directly with the square root of temperature. In liquids, the momentum exchange due to molecular movement is small compared to the cohesive forces between the molecules. Thus, the viscosity is primarily dependent on the magnitude of these cohesive forces. Since these forces decrease rapidly with increases of temperature, liquid viscosities decrease as temperature increases.

1.5. Surface tension

Surface tension, $(\sigma)$, is a material property whereby a liquid at a material interface, usually liquid-gas, exerts a force per unit length along the surface. According to the theory of molecular attraction, molecules of liquid considerably below the surface act on each other by forces that are equal in all directions. However, molecules near the surface have a greater attraction for each other than they do for molecules below the surface because of the presence of a different substance above the surface. This produces a layer of surface molecules on the liquid that acts like a stretched membrane. Because of this membrane effect, each portion of the liquid surface exerts “tension” on adjacent portions of the surface or on objects that are in contact with the liquid surface. This tension acts in the plane of the surface, and is given by:

$$F = \sigma L \rightarrow \sigma = \frac{F}{L}$$  \hspace{1cm} (1.5.1)

where $L$ is the length over which the surface tension acts. This force is very slightly influenced by what the surrounding gas is air or water vapor or some other gas. Typical values of the surface tension of liquid surfaces exposed to air are shown in Table 1.5.
Table 1.5. Surface tension of pure fluids exposed to air at 20°C.

<table>
<thead>
<tr>
<th>Fluid</th>
<th>Surface tension lbf/in</th>
</tr>
</thead>
<tbody>
<tr>
<td>Acetic acid</td>
<td>0.00016</td>
</tr>
<tr>
<td>Acetone</td>
<td>0.00014</td>
</tr>
<tr>
<td>Benzene</td>
<td>0.00016</td>
</tr>
<tr>
<td>Carbon tetrachloride</td>
<td>0.00015</td>
</tr>
<tr>
<td>Ethyl alcohol</td>
<td>0.00013</td>
</tr>
<tr>
<td>n-Octane</td>
<td>0.00012</td>
</tr>
<tr>
<td>Toluene</td>
<td>0.00016</td>
</tr>
<tr>
<td>Water</td>
<td>0.00043</td>
</tr>
<tr>
<td>Mercury</td>
<td>0.0030</td>
</tr>
</tbody>
</table>

For all liquids the surface tension decreases as the temperature rises. The surface tension of water may be considerably reduced by the addition of small quantities of organic solutes such as soap and detergents. Salts such as sodium chloride in solution raise the surface tension of water. That tension which exists in the surface separating two immiscible liquids is usually known as interfacial tension.

As a consequence of surface tension effects a drop of liquid, free from all other forces, takes on a spherical form. The molecules of a liquid are bound to one another by forces of molecular attraction, and it is these forces that give rise to cohesion, that is, the tendency of the liquid to remain as one assemblage of particles rather than to behave as a gas and fill the entire space within which it is confined. Forces between the molecules of a fluid and the molecules of a solid boundary surface give rise to adhesion between the fluid and the boundary.

If the forces of adhesion between the molecules of a particular liquid and a particular solid are greater than the forces of cohesion among the liquid molecules themselves, the liquid molecules tend to crowd towards the solid surface, and the area of contact between liquid and solid tends to increase. Given the opportunity, the liquid then spreads over the solid surface and ‘wets’ it. Water will wet clean glass, but mercury will not. Water, however, will not wet wax or a greasy surface.

1.6. Newtonian and non-Newtonian fluids

Fluids for which the shear stress is directly proportional to the rate of strain are called Newtonian fluids. Because shear stress is directly proportional to the shear strain, a plot relating these variables (see Fig. 1.6.1) results in a straight line passing through the origin. The slope of this line is the value of the dynamic viscosity.

Newtonian fluids are described in another way as fluids in which the tangential stresses are linearly proportional to the velocity gradients. Between two parallel plates is filled with a Newtonian fluid. When the upper plate is moved with the velocity \( u_w \) parallel the lower plate, as given in the Fig. 1.6.2, then the velocity increases linearly in the \( y \)-direction. The particles in the superjacent layers move with different velocities:

\[ u_y = u_w \frac{y}{h} \]  

(1.6.1)
Fluids which do not follow the linear law of Equation (1.6.1) are called Non-Newtonian. One class of non-Newtonian fluids, shear-thinning fluids, has the interesting property that the ratio of shear stress to shear strain decreases as the shear strain increases (see Figure 1.6.1). Some common shear-thinning fluids are toothpaste, catsup, paints, and printer’s ink. Fluids for which the viscosity increases with shear rate are shear-thickening fluids. Some examples of these fluids are mixtures of glass particles in water and gypsum-water mixtures. Another type of non-Newtonian fluid, called a Bingham plastic, acts like a solid for small values of shear stress and then behaves as a fluid at higher shear stress. The shear stress versus shear strain rate for a Bingham plastic is also shown in Fig. 1.6.1.

The Bingham model describes the flow process of fluids, which below a certain shear stress behave as a rigid body:

\[
\tau = \mu \dot{\gamma} \pm \tau_o \tag{1.6.2}
\]
2. Fundamentals of fluid kinematics and dynamics

Yadullah Babayev, Orkhan Huseyinov
Engineering Faculty, Qafqaz University
2.1. Conservation laws for the motion

Fields such as density field \( \rho(x,t) \), pressure field \( p(x,t) \), velocity field \( v(x,t) \), temperature field \( T(x,t) \) etc. are used to represent the fluid flows. The \( x \) and \( t \) variables in fluid fields define their values at a point \( x \) and time \( t \). In the Cartesian frame of reference of \( (x,y,z) \) or in the similar way as \( (x_1,x_2,x_3) \) are used to describe the position vector \( x \). The velocity of fluid particles are represented as \( \frac{dx}{dt} = v(x,t) \).

Fundamental conservation laws of physics are used to show the evolution of fluid flow with respect to time. There are three types of conservation laws of mechanics which are conservation law of mass, momentum and energy. The homogeneity and isotropy of space and time are the fundamental causes of the conservation laws. The conservation of energy is derived from the homogeneity of time in the Lagrangian function of a closed system. However, from the homogeneity of space, the conservation of momentum is achieved. Isotropy of space is the cause of conservation of angular momentum. Conservation of mass comes from the invariance of the relativistic Lagrangian in the Newtonian limit (Kambe, 2007).

2.1.1. Laws of mass, momentum and energy conservation

Conservation law of mass. In this section, the partial differential equations will be derived for representing conservation of mass, momentum and energy in a fluid flow in other words the continuity equations will be achieved. At the end some specific examples will be given to employ the equations for solving problems.

Mass inside the control volume fixed in the space with a surface \( S \) and volume \( V \) is shown by \( \int \rho dV \), so the

\[
\text{Decrease rate of mass in volume (V)} = -\frac{d}{dt} \int \rho dV = -\int \frac{\partial \rho}{\partial t} dV \tag{2.1.1}
\]

As the mass is conserved, the overall rate of mass flux (\( m \)- rate of mass while crossing the control surface) out of \( V \) must equal to equation 1. The rate of mass flux out of \( V \) across any small element \( dS \) of \( S \) is \( \rho v dS \) and the dimension of \( dS \) is identical to the elements area. \( dS \) is taken along the outward normal. After the total surface is integrated, the results are as follows:

\[
\text{Rate of mass flux out of } V = \int_S \rho v dS = \int \nabla \cdot \rho v dV \tag{2.1.2}
\]

Integrand (\( \nabla \)) on the right hand side of the equation is shown in Cartesian coordinates \( x=(x, y, z) \) and \( v=(u, v, w) \) as

\[
\nabla \cdot \rho v = \frac{\partial (\rho u)}{\partial x} + \frac{\partial (\rho v)}{\partial y} + \frac{\partial (\rho w)}{\partial z} \tag{2.1.3}
\]
For law of mass conservation, equations 1 and 2 must be identical i.e,

**Continuity equation:**
\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{v}) = 0
\]  
(2.1.4)

By continuity equation, the change rate of any physical quantity \( f = f(x, t) \) at a defined point which is fixed in the space is taken into account, however if change rate of \( f \) in a given element of fluid is required along the trajectory of \( x = x(t) \), material derivative is obtained as below;

\[
\frac{\partial f}{\partial t} = \frac{df}{dt} = f_{,t} + \mathbf{v} \cdot \nabla f = \frac{\partial f}{\partial t} + \frac{\partial f}{\partial x} \frac{dx}{dt} + \frac{\partial f}{\partial y} \frac{dy}{dt} + \frac{\partial f}{\partial z} \frac{dz}{dt} = \frac{\partial f}{\partial t} + \mathbf{v} \cdot \nabla f
\]  
(2.1.5)

In case of incompressible fluid, the continuity equation equals to \( \nabla \cdot \mathbf{V} = 0 \), since the density in incompressible fluid is constant in any space and time - \( \frac{D\rho}{Dt} = 0 \). In Cartesian coordinates the equation is;

\[
\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0
\]  
(2.1.6)

**Conservation law of momentum.** There are two different types of forces on any part of the fluid with regards to spatial scales. Long-range forces influence all the parts of the fluid having a macroscopic range and called volume forces as they are mostly proportional to the fluid volume. The other kind of force has microscopic range and is proportional to the surface area; therefore these forces are called surface forces. The mathematical derivation of surface forces is attributed to Cauchy. These forces can be generalized as (Olbers et al., 2012)

\[
= \nu \, dV \rho g + s[\Pi] \cdot dS = \nu \, dV (\rho g + v \cdot [\Pi])
\]  
(2.1.7)

where \( \Pi \) – stress tensor; \( dV \rho g \) – gravity as an external force; \( dS \) – force acting on unit area across a surface element.

If a volume \( V \) with a material surface of \( S \) moves with the flow always including the same material elements, its momentum is,
\[ \dot{\rho} dV \rho v \]  

Then:

\[
\frac{d}{dt} \rho dV \rho v = \rho dV \frac{\dot{\rho}}{dt} \tag{2.1.9}
\]

The mass of each element \( \rho dV \) is constant and has to be equal to the whole forces exerting on it. Therefore, equations 2.1.7 and 2.1.9 must be equal according to Newton’s second law. Finally, Cauchy equation is achieved:

\[
\rho \frac{\dot{V}}{dt} = \rho g + \nabla \tag{2.1.10}
\]

If it is written in Cartesian coordinates, it will be:

**Momentum, x:**

\[
\rho \frac{\dot{V}}{dt} = \rho g_x + \frac{\partial}{\partial x} \Pi_{xx} + \frac{\partial}{\partial y} \Pi_{xy} + \frac{\partial}{\partial z} \Pi_{xz}
\]

**Momentum, y:**

\[
\rho \frac{\dot{V}}{dt} = \rho g_y + \frac{\partial}{\partial x} \Pi_{yx} + \frac{\partial}{\partial y} \Pi_{yy} + \frac{\partial}{\partial z} \Pi_{yz}
\]

**Momentum, z:**

\[
\rho \frac{\dot{V}}{dt} = \rho g_z + \frac{\partial}{\partial x} \Pi_{xz} + \frac{\partial}{\partial y} \Pi_{zy} + \frac{\partial}{\partial z} \Pi_{zz}
\]

![Figure 2.1.2. Surface stress in fluid element in 2D](image)

**The conservation law of energy** is the specific application of the first law of fluid dynamics. The amount of heat released per unit volume is \( \dot{\phi} = \rho c_p T \) for a single phase material (\( c_p \) is lower calorific value at constant pressure while \( T \) represents the temperature). If there is no transport, the heat flux is defined as:

\[
F = -k \nabla T \tag{2.1.12}
\]

where \( k \) – thermal conductivity.

As heat flows from hot to cold, \( \nabla T \) is negative. The transport flux is \( \rho c_p T \nabla V \). Heat can be created in a region because of radioactive decay, viscous dissipation etc. (all these sources will be defined as \( H \) in the equation) however mass cannot. Simplest conservation of heat equation is:

\[
\frac{\partial (\rho c_p T V)}{\partial t} + \nabla (\rho c_p T V) = \nabla k \nabla T + H \tag{2.1.13}
\]
For constant \( c_p \) and \( k \), the equation can be rearranged as:

\[
\frac{\partial T}{\partial t} + \nabla \cdot \text{V} \nabla T = \kappa \nabla^2 T + H/\rho c_p
\]

(2.1.14)

where \( \kappa = k/\rho c_p \) is the thermal diffusivity.

2.1.2. Basic equations: continuity equation, equations of motion and energy

In a time dependent volume \( \tau(t) \) the mass of a flowing medium bounded with closed surface \( A(t) \) is constant:

\[
\frac{dm}{dt} = \frac{d}{dt} \int_{\tau(t)} \rho d\tau = 0
\]

(2.1.15)

After calculating the total time derivative of the equation above:

\[
\frac{d}{dt} \int_{\tau(t)} \rho d\tau = \lim_{\Delta t \to 0} \frac{1}{\Delta t} \left[ \int_{\tau(t)+\Delta t} \rho \ d\tau - \int_{\tau(t)} \rho \ d\tau \right]
\]

(2.1.16)

With \( \rho \ t + \Delta t = \rho \ t + \frac{\partial \rho}{\partial t} \Delta t + \ldots \) and \( d\tau = (v \cdot n) dA \ dt \)

(2.1.17)

Then they result in:

\[
\frac{d}{dt} \int_{\tau(t)} \rho d\tau = \int_{\tau(t)} \frac{\partial \rho}{\partial t} d\tau + \int_{A(t)} \rho \ v \ n \ dA
\]

(2.1.18)

Afterwards, using Gauss divergence theorem, the surface integral over \( A(t) \) is transformed into a volume integral as shown in equation below;

\[
\frac{d}{dt} \int_{\tau(t)} \rho d\tau = \int_{\tau(t)} \frac{\partial \rho}{\partial t} d\tau + \nabla \cdot (\rho \ v) \ dt
\]

(2.1.19)

The integrand is vanished throughout the flow field to achieve the volume integral above vanished. After that new equation derived is called continuity equation;

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \ v) = 0
\]

(2.1.20)

The equation above can be shown in other form after introducing the total time derivative of density;

\[
\frac{\partial \rho}{\partial t} + \rho (\nabla \cdot v) = 0
\]

(2.1.21)

where \( \nabla \cdot v = 0 \) in incompressible flow.
It is also possible to derive the continuity equation with the aid of a mass balance for a space-bound volume element. The difference between in and outward flowing mass is equal to the change of mass per unit time in the volume element:

$$\frac{\partial (\rho u)}{\partial x} \, dx \, dy \, dz + \frac{\partial (\rho v)}{\partial y} \, dx \, dy \, dz + \frac{\partial (\rho w)}{\partial z} \, dx \, dy \, dz = \frac{\partial \rho}{\partial t} \, dx \, dy \, dz$$

(2.1.22)

The same equation is obtained at the end:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot \rho \nu = 0$$

(2.1.23)

where $\frac{\partial \rho}{\partial t}$ is the change of mass per unit time in the control volume and $\nabla, \rho, \nu$ is the mass flow through its surface (Krause, 2005).

### 2.2. Generalized Newton law of viscous friction

Viscosity is the characteristic feature of some fluids and in a general definition; viscosity is the resistance to shear stresses. Viscosity is something like “internal friction”. However it seems contradictory to the definition of fluids that they do not have resistance to deformation because of shear stresses, it simply illustrates the limited rate of deformation or in other words the varying resistance degree of fluids to shear stress (compare honey, tar and water at same temperature). These different degrees can be formulated as the Newton’s law of viscosity. Newton’s law of viscosity shows that shear stress is directly proportional to viscosity in a given rate of angular deformation.

If the tangential stresses of fluids are linearly proportional to the velocity gradients, they are called Newtonian fluids. The dependence can be illustrated in this experiment. There are two parallel plates and the space between them is filled with a Newtonian fluid.
As described in the figure, there are two plates located in a distance between each other. When a tangential force is applied to the upper plate to move it parallel to the lower plate with the velocity of \( v_w \) in the x direction, then the velocity will rise in the y-direction. This force to move the upper plate is proportional to the speed of that plate and its area, on the contrary inversely proportional to the distance between these plates:

\[
F = \mu \cdot \frac{AU}{h} \tag{2.2.1}
\]

As the average shear stress is represented by force over area, then the equation above becomes

\[
\text{Average shear stress} = \mu \cdot \frac{v}{h} \tag{2.2.2}
\]

\( U/h \) is the angular velocity of line a-b in Figure 2.2.1 and the rate of angular deformation of the fluid. That formula is Newton’s law of viscosity. To apply the formula at any point of fluid, the formula can be differentiated as:

\[
\tau = \mu \frac{du}{dy} \tag{2.2.3}
\]

Meanwhile, the particles in the overlying layers will move with various velocities. When \( y=0 \), the velocity is zero and \( y= h \) it is \( v_w \) (McDonough, 2009):

\[
u \gamma = u_w \frac{\gamma}{h} \tag{2.2.4}
\]

Then the angle after shear stress can be defined in the following way:

\[
\Delta \gamma \approx -\frac{u_w \Delta t}{h} \tag{2.2.5}
\]

After forming the differential quotient the rate of strain is achieved:

\[
\dot{\gamma} = \lim_{\Delta t \to 0} \frac{\Delta \gamma}{\Delta t} = -\frac{u_w}{h} \tag{2.2.6}
\]

General way for showing the rate of strain for velocity distributions is as:

\[
\gamma = -\frac{du}{dy} \tag{2.2.7}
\]

The relation between the shear stress and the rate of strain is proportional and to show the constant of proportionality dynamic shear viscosity \( \mu \) is used. The pressure, temperature and medium are the main factors influencing \( \mu \). The ratio of \( \mu \) over density \( \rho \) is called kinematic shear viscosity, i.e,

\[
\nu = \frac{\mu}{\rho} \tag{2.2.8}
\]
The definitions of the symbols and equations are given as follows:

- Rigid body: \( \tau = f(\gamma) \)
- Fluid: \( \tau = f(y) \)
- \( \gamma \): Angle of shear stress
- \( y \): Rate of strain
- \( \tau = G \cdot \gamma \)
- Newton’s viscosity law: \( \tau = \mu \cdot \dot{\gamma} \)

In parallel shear flow as described in Figure 2.2.2, there is friction flow acting on the plate because of the flow. To represent the friction force per unit area, the equation below is used:

\[
\sigma_f = \mu \frac{du}{dy} \bigg|_{y=0} \tag{2.2.9}
\]

where \( \mu \) is the coefficient of shear viscosity and is called Newton’s law of viscosity (Kambe, 2007).

### 2.3. Motion equations for viscous continuum: Navier-Stokes equations

Navier-Stokes equations were firstly discovered by Navier in 1821 and after him, a lot of contributions have been made to the equations by ignoring or adding some parts with their own way to justify it (Kambe, 2007).

As mentioned before the time rate of change of momentum of a closed system is equal to the total external forces influencing the volume:

\[
\frac{d}{dt} \tau(t) \rho v d\tau = F \tag{2.3.1}
\]

After rearranging the equation above:

\[
\frac{d}{dt} \tau(t) \rho v d\tau = \tau(t) \frac{\partial \rho v}{\partial t} + \nabla \rho v v \ d\tau = \tau(t) \rho \frac{\partial v}{\partial t} + v \cdot \nabla v \ d\tau \tag{2.3.2}
\]

The \( v \) is the dyadic (combination of two vectors) product of the vector \( v \). The gravitational force:

\[
F_g = \tau(t) \rho g d\tau \tag{2.3.3}
\]

And the surface forces:

\[
F_\sigma = - A(t) \ n \cdot \sigma \ dA = - \tau(t) \ \nabla \cdot \sigma \ d\tau \tag{2.3.4}
\]

where \( n \) – the normal vector, \( \sigma \) – stress tensor.

There can be two examples of external forces.
For an arbitrarily closed volume the momentum theorem is:

\[
\tau(t) \rho \frac{\partial v}{\partial t} + v \cdot \nabla v \, dt = \tau(t) \rho g \, dt - \tau(t) \nabla \sigma \, dt \quad (2.3.5)
\]

\[
\rho \frac{\partial v}{\partial t} + v \cdot \nabla v = \rho g - (\nabla \sigma) \quad (2.3.6)
\]

In Cartesian coordinates:

\[
\sigma = \begin{pmatrix} \sigma_{xx} & \tau_{xy} & \tau_{xz} \\ \tau_{yx} & \sigma_{yy} & \tau_{yz} \\ \tau_{zx} & \tau_{zy} & \sigma_{zz} \end{pmatrix}
\]

\[
\rho \frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} = \partial g_x - \frac{\partial \sigma_{xx}}{\partial x} - \frac{\partial \tau_{xy}}{\partial y} - \frac{\partial \tau_{xz}}{\partial z}
\]

\[
\rho \frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + w \frac{\partial v}{\partial z} = \partial g_y - \frac{\partial \sigma_{yy}}{\partial y} - \frac{\partial \tau_{yx}}{\partial x} - \frac{\partial \tau_{yz}}{\partial z}
\]

\[
\rho \frac{\partial w}{\partial t} + u \frac{\partial w}{\partial x} + v \frac{\partial w}{\partial y} + w \frac{\partial w}{\partial z} = \partial g_z - \frac{\partial \sigma_{zz}}{\partial z} - \frac{\partial \tau_{zx}}{\partial x} - \frac{\partial \tau_{zy}}{\partial y} \quad (2.3.7)
\]

The equations above show the relations between the local stress of the fluid and its velocity components. From these equations, it is clear that as in the second law of Newton, a small volume element moving with fluid is decelerated or accelerated by the external forces affecting it.

In Hooke’s law it is stated that stresses are proportional to the strain however in fluid mechanics the stress are proportional to the time rate change of the strain. To illustrate the relation between them, it assumed that shear stresses \( \tau_{xy}, \ldots, \tau_{xy} \) cause angular displacements \( \gamma_{xy}, \ldots, \gamma_{xy} \) and normal stresses \( \sigma_{xx}, \sigma_{yy}, \sigma_{zz} \) result in elongations and contractions \( \varepsilon_x, \varepsilon_y, \varepsilon_z \).

\[\varepsilon_x = \frac{\partial u}{\partial x} \quad \varepsilon_y = \frac{\partial v}{\partial y} \quad \varepsilon_z = \frac{\partial w}{\partial z} \quad (2.3.8)\]

\[e = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = \nabla \cdot v\]

The angular displacements per time interval \( dt \) are:

\[\gamma_{xy} = \frac{d\beta + d\alpha}{dt} = -\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\]
Stokes introduced a hypothesis that in the state of rest, time rate of change of angular displacements and strains vanish, normal stresses $\sigma_{xx}$, $\sigma_{yy}$, $\sigma_{zz}$ are not dependent on the direction and described by hydrostatic pressure:

\[
\sigma_{xx} = p - 2\mu \epsilon_{x} - \lambda \epsilon
\]

\[
\sigma_{yy} = p - 2\mu \epsilon_{y} - \lambda \epsilon
\]

\[
\sigma_{zz} = p - 2\mu \epsilon_{z} - \lambda \epsilon
\]

And

\[
\tau_{xy} = \mu \gamma_{xy}
\]

\[
\tau_{yz} = \mu \gamma_{yz}
\]

\[
\tau_{xz} = \mu \gamma_{xz}
\]

\[
\sigma = \begin{pmatrix} 1 & -\lambda & \nabla \cdot \mathbf{v} \\ -\lambda & 1 & \nabla \cdot \mathbf{v} \\ \nabla \cdot \mathbf{v} & \nabla \cdot \mathbf{v} & -2\mu \begin{pmatrix} \frac{1}{2} \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} \\ \frac{1}{2} \frac{\partial u}{\partial x} + \frac{\partial w}{\partial z} \\ \frac{1}{2} \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} \end{pmatrix} \end{pmatrix}
\]

The coefficient of $\lambda$ is equal to $\mu - \frac{2}{3} \mu$. For monoatomic gases the volume viscosity of $\mu$ vanishes. Equations for normal stresses:

\[
\sigma_{xx} = p - 2\mu \frac{\partial u}{\partial x} - \mu - \frac{2}{3} \mu \nabla \cdot \mathbf{v}
\]

\[
\sigma_{yy} = p - 2\mu \frac{\partial v}{\partial y} - \mu - \frac{2}{3} \mu \nabla \cdot \mathbf{v}
\]

\[
\sigma_{zz} = p - 2\mu \frac{\partial w}{\partial z} - \mu - \frac{2}{3} \mu \nabla \cdot \mathbf{v}
\]

The mean value is

\[
\sigma = \frac{1}{3} \sigma_{xx} + \sigma_{yy} + \sigma_{zz} = p - \mu \nabla \cdot \mathbf{v}
\]

where $\sigma = p$ for incompressible flows.

\[
\rho \frac{du}{dt} = -\frac{\partial p}{\partial x} + \frac{\partial}{\partial x} 2\mu \frac{\partial u}{\partial x} + \lambda \nabla \cdot \mathbf{v} + \frac{\partial}{\partial y} \mu \frac{\partial u}{\partial y} + \frac{\partial}{\partial z} \mu \frac{\partial u}{\partial z} + \rho g_x
\]

\[
\rho \frac{dv}{dt} = -\frac{\partial p}{\partial y} + \frac{\partial}{\partial y} 2\mu \frac{\partial v}{\partial y} + \lambda \nabla \cdot \mathbf{v} + \frac{\partial}{\partial z} \mu \frac{\partial v}{\partial z} + \rho g_y
\]

\[
\rho \frac{dw}{dt} = -\frac{\partial p}{\partial z} + \frac{\partial}{\partial z} 2\mu \frac{\partial w}{\partial z} + \lambda \nabla \cdot \mathbf{v} + \rho g_z
\]

After substituting the stress-strain relations Navier-Stokes equations are obtained:
As we mentioned, \( \lambda = \mu - \frac{2}{3} \mu \), \( \mu \) is not taken account the Navier-Stokes equations are getting the form below:

\[
\rho \frac{Du}{Dt} = \rho g_x - \frac{\partial p}{\partial x} + \frac{\partial}{\partial x} \mu \frac{2}{3} \nabla V + \frac{\partial}{\partial y} \mu \frac{2}{3} \nabla V + \frac{\partial}{\partial z} \mu \frac{2}{3} \nabla V + \frac{\partial}{\partial x} \mu \frac{2}{3} \nabla V + \frac{\partial}{\partial y} \mu \frac{2}{3} \nabla V + \frac{\partial}{\partial z} \mu \frac{2}{3} \nabla V
\]

\[
\rho \frac{Dv}{Dt} = \rho g_y - \frac{\partial p}{\partial y} + \frac{\partial}{\partial y} \mu \frac{2}{3} \nabla V + \frac{\partial}{\partial x} \mu \frac{2}{3} \nabla V + \frac{\partial}{\partial z} \mu \frac{2}{3} \nabla V + \frac{\partial}{\partial x} \mu \frac{2}{3} \nabla V + \frac{\partial}{\partial y} \mu \frac{2}{3} \nabla V + \frac{\partial}{\partial z} \mu \frac{2}{3} \nabla V
\]

\[
\rho \frac{Dw}{Dt} = \rho g_z - \frac{\partial p}{\partial z} + \frac{\partial}{\partial z} \mu \frac{2}{3} \nabla V + \frac{\partial}{\partial x} \mu \frac{2}{3} \nabla V + \frac{\partial}{\partial y} \mu \frac{2}{3} \nabla V + \frac{\partial}{\partial x} \mu \frac{2}{3} \nabla V + \frac{\partial}{\partial y} \mu \frac{2}{3} \nabla V + \frac{\partial}{\partial z} \mu \frac{2}{3} \nabla V
\]

The above equations decrease for the incompressible flows having constant dynamic shear viscosity:

\[
\rho \frac{du}{dt} = -\frac{\partial p}{\partial x} + \mu \nabla^2 u + \rho g_x
\]

\[
\rho \frac{dv}{dt} = -\frac{\partial p}{\partial x} + \mu \nabla^2 u + \rho g_y
\]

\[
\rho \frac{dw}{dt} = -\frac{\partial p}{\partial x} + \mu \nabla^2 u + \rho g_z
\]
3. The Bernoulli equations
3.1. Euler’s equation of motion

Navier-Stokes equation was described by Euler neglecting the viscosity of fluids. However all fluids do possess viscosity, inviscid flow actually represents the fluid flows of which total viscous forces are very small compared to inertial forces or pressure. Inviscid flows have high Reynolds number. $\mu \nabla^2 V$ term is neglected in Navier-Stokes equation while defining Euler equation and that term includes highest-order derivatives of velocity. Loss of the term $\mu \nabla^2 V$ will decrease number of boundary conditions. When Euler equations are used, the no-slip boundary condition cannot be specified at solid walls and therefore results of Euler equations are not meaningful near the solid walls as shown in figure 3.1.1.

![Euler equation valid](image1)

![Euler equation not valid](image2)

Figure 3.1.1. Euler equation is only appropriate in regions of high Reynolds number and total smaller viscous forces compared to inertial and pressure forces (Cengel, 2013)

After writing the general Navier-Stokes equations

$$\rho \frac{du}{dt} = \rho g_x - \frac{\partial p}{\partial x} + \frac{\partial}{\partial x} \left( \mu \left( \frac{\partial u}{\partial x} + \frac{\partial u}{\partial y} \right) \frac{2}{3} \nabla V \right) + \frac{\partial}{\partial y} \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right) + \frac{\partial}{\partial z} \left( \frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} \right)$$

$$\rho \frac{dv}{dt} = \rho g_y - \frac{\partial p}{\partial y} + \frac{\partial}{\partial x} \left( \mu \left( \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} \right) \frac{2}{3} \nabla V \right) + \frac{\partial}{\partial y} \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial y} \right) + \frac{\partial}{\partial z} \left( \frac{\partial u}{\partial z} + \frac{\partial w}{\partial y} \right)$$

$$\rho \frac{dw}{dt} = \rho g_z - \frac{\partial p}{\partial z} + \frac{\partial}{\partial x} \left( \mu \left( \frac{\partial u}{\partial x} + \frac{\partial w}{\partial z} \right) \frac{2}{3} \nabla V \right) + \frac{\partial}{\partial y} \left( \frac{\partial u}{\partial y} + \frac{\partial w}{\partial y} \right) + \frac{\partial}{\partial z} \left( \frac{\partial u}{\partial z} + \frac{\partial w}{\partial z} \right)$$

(3.1.1)

(Pritchard, 2011)

The three components of Euler equations will then be written in Cartesian coordinates as below

$$\rho \frac{du}{dt} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} = \rho g_x - \frac{\partial p}{\partial x}$$

$$\rho \frac{dv}{dt} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + w \frac{\partial v}{\partial z} = \rho g_y - \frac{\partial p}{\partial y}$$

$$\rho \frac{dw}{dt} + u \frac{\partial w}{\partial x} + v \frac{\partial w}{\partial y} + w \frac{\partial w}{\partial z} = \rho g_z - \frac{\partial p}{\partial z}$$

(3.1.2)

(Shaughnessy, 2005)

In order to gain the flow of an ideal fluid, Euler’s equation of motion and continuity equation must be solved under the given boundary and initial conditions. Three quantities of $u$, $v$ and $p$ as the functions of $t$ and $x$, $y$ are to be calculated. The calculation is gained for a particular restricted case as the acceleration term is non-linear (Nakayama, 2000).
3.2. Bernoulli equation for ideal flow

As ideal flow is taken into account to be the case for calculating Bernoulli equation, the properties of ideal fluid need to be defined:

1. Element of an ideal fluid cannot change its volume while moving i.e., it is incompressible.
2. All fluid elements have same density for all time i.e., it has constant density.
3. Ideal fluids are inviscid.
4. Ideal fluids are irrotational, there is no turbulence (Zielinski, 2012).

However there is no fully ideal fluid in nature.

To obtain Bernoulli equation for fluid, Newton’s second law is applied to a fluid element along a streamline. The Bernoulli equation gives the possible relation among velocity, pressure and elevation. And the equation is valid in the zones of steady (there is no change at a defined location while time alters) and incompressible flow of which total frictional forces are neglected. Bernoulli equation is only approximation applied to only inviscid regions.

As shown in Figure 3.2.1. the motion of a fluid particle is illustrated as the distances along a streamline along with the radius of curvature through the streamline. Velocity of particle is

\[ V = \frac{ds}{dt} \] (3.2.1)

The velocity is chosen to be the function of s and t. Then,

\[ dV = \frac{\partial V}{\partial s} ds + \frac{\partial V}{\partial t} dt \quad \text{and} \quad \frac{dV}{dt} = \frac{\partial V}{\partial s} \frac{ds}{dt} + \frac{\partial V}{\partial t} \] (3.2.2)

where \(\frac{\partial V}{\partial t}\) is zero in steady flow and the acceleration in the s-direction in Figure 3.2.1. becomes:

\[ a_s = \frac{dV}{dt} = \frac{\partial V}{\partial s} \frac{ds}{dt} = \frac{\partial V}{\partial s} V = V \frac{dV}{ds} \] (3.2.3)
Steady flow has acceleration since there is change from one position to the next one. Applying Newton’s second law (in fluid mechanics conservation of linear momentum)

\[ F_s = ma_s \]  
(3.2.4)

As the fluid is inviscid, forces in the s-direction are the component of the weight of the particle and pressure. Then Equation (3.2.4) changes to:

\[ PdA - (P+dP)dA - Wsin\theta = mV_{av}^{dv/ds} \]  
(3.2.5)

where \( m=\rho V = \rho dA \) ds is mass; \( W=mg=\rho gdA \) ds is weight of the fluid particle; \( \theta \) shows the angle between normal of the streamline and the vertical z-axis at that place and \( sin\theta= dz/ds \). Afterwards,

\[-dP.dA - \rho g.dA \frac{dx}{ds} = \rho dAdV_{av}^{dv/ds} \]  
(3.2.6)

If \( dA \) is cancelled,

\[-dP - \rho gd\frac{dx}{ds} = \rho dsV_{av}^{dv/ds} \]  
(3.2.7)

As \( VdV=\frac{1}{2}d(V^2) \) and after Equation (3.2.7) is devided by \( \rho \), it simplifies:

\[ \frac{dp}{\rho} + \frac{1}{2}d(V^2) + g dz = 0 \]  
(3.2.8)

Finally Bernoulli equations for steady flow and steady, incompressible flow are achieved by integrating equations.

For steady flow:

\[ \frac{dp}{\rho} + \frac{V^2}{2} + gz=\text{constant along the streamline} \]  
(3.2.9)

For steady, incompressible flow:

\[ \frac{p}{\rho} + \frac{V^2}{2} + gz=\text{constant along the streamline} \]  
(3.2.10)

(Cengel, 2013)

The Equations (3.2.9) and (3.2.10) are known Bernoulli equations and equation (3.2.10) is the Bernoulli equation for the ideal fluids.

### 3.3. Bernoulli equation for real flow

Bernoulli equation is applied to non-viscous fluids, however all real fluids have viscosity. An external force is needed to apply to maintain the flow in viscous fluid, especially maintaining the pressure difference at the ends of the pipe.

![Figure 3.3.1. Net force applied in the counter-direction to flow](Fv-viscous force) (Giambattista, 2009)
In real fluid flow two losses can be seen:

1. The linear loss head-causing from the friction between the fluid and walls of the flow;
2. The local loss head- in the case of valves, forks etc.

![Figure 3.3.2. Real fluid flow particle with forces acting on it](image)

The derivation of the equation for the real fluids is starting from:

\[ F = ma, \]

(3.3.1)

If we substitute \( dA \) with \( \pi r^2 \) and if divide each term by \( (p \cdot \pi r^2) \), the equation will become:

\[ \frac{dp}{\rho} + V. dv + g. dz = -\frac{2\pi \cdot ds}{\rho \cdot r} \]  

(3.3.2)

which is the Euler’s equation for real fluids. After dividing each term in Euler’s equation of motion with \( g \) and integrating the equation, Bernoulli equation will be achieved as below:

\[ \frac{2}{\gamma} \frac{dp}{d\gamma} + \frac{2}{\gamma} \frac{V^2}{2g} + \frac{2}{\gamma} dz = \frac{2}{\gamma} \frac{2\pi \cdot ds}{\gamma \cdot r} \]

(3.3.3)

\[ \frac{p_2}{\gamma} - \frac{p_1}{\gamma} + \frac{V_2^2}{2g} - \frac{V_1^2}{2g} + z_2 - z_1 = -\frac{2\pi L}{\gamma \cdot r} \]

where \( \frac{2\pi L}{\gamma \cdot r} \) is shown as \( h_L \) of energy loss, then the general form of Bernoulli equation is:

\[ \frac{p_1}{\gamma} + \frac{V_1^2}{2g} + z_1 = \frac{p_2}{\gamma} + \frac{V_2^2}{2g} + z_2 + h_L \]  

(3.3.4)

(Korukcu, 2003)
3.4. Various forms of Bernoulli equations

The control volume has boundaries that is not possible to penetrate inside except that the inlets or outlets through which fluid enters and leaves the volume. When fluid passes, the heat is obtained by the irreversible transformation of mechanical energy and it leads to losses. This can also lead to “losses” due to friction. On the other hand, work performed by fluid or on fluid can result in “shaft work” (if it is by fluid, it is assumed by convention to be positive otherwise it is negative).

Bernoulli equations has two common forms with a single fluid inlet and outlet-energy form and head form (Subramanian, 2010).

3.4.1. Energy form

In the energy form of the Bernoulli equation each term has its dimension of energy per unit mass.

\[
\frac{p_\text{out}}{\rho} + \frac{v_{\text{out}}^2}{2} + g z_{\text{out}} = \frac{p_\text{in}}{\rho} + \frac{v_{\text{in}}^2}{2} + g z_{\text{in}} - \text{loss} - w_s
\]

where \(z\) – elevation above a specified datum; \(g\) – acceleration; \(\text{loss}\) – losses per unit mass flowing through; \(w_s\) – shaft work performed by fluid per unit mass flowing through. Loss can be as a certain (M) head of fluid, i.e., loss=\(gM\) or number of velocity heads\(^1\), i.e. loss=\(N\ (V^2/2)\).

Pressure needs to be in the same basis, absolute or gauge, on both sides. The equation above is true for turbulent flow as velocity profiles across inlet and outlet are flat, however in laminar flow average velocities are taken at the inlet and outlet and kinetic energy terms in both sides of the equation (\(\frac{v_{\text{out}}^2}{2}, \frac{v_{\text{in}}^2}{2}\)) are multiplied by the correction factor \(\alpha\).

3.4.2. Head form

Head form of the Bernoulli equation is attained by dividing the energy terms of the equation by acceleration;

\[
\frac{p_\text{out}}{\gamma} + \frac{v_{\text{out}}^2}{2g} + z_{\text{out}} = \frac{p_\text{in}}{\gamma} + \frac{v_{\text{in}}^2}{2g} + z_{\text{in}} - \frac{\text{loss}}{g} - \frac{w_s}{g}
\]

where \(\gamma=\rho \cdot g\) and shows the specific weight of fluid.

Head developed by pump is always positive and the formula is \(h_p = -\frac{w_s}{g}\). Work term of \(w_s\) is negative for the pump. On the other hand loss term is positive and represented as

\[
\frac{\text{loss}}{g} = h_{\text{friction}} = h_f
\]

\(^1\)Kinetic energy of a fluid flow within a given streamtube and it is measured by the elevation which would be reached in the upward direction of velocity.
Finally retyping the head form of the Bernoulli equation (Subramanian, 2010):

\[
\frac{p_{\text{out}}}{\gamma} + \frac{v_{\text{out}}^2}{2g} + z_{\text{out}} = \frac{p_{\text{in}}}{\gamma} + \frac{v_{\text{in}}^2}{2g} + z_{\text{in}} - h_f - h_p
\]  (3.4.4)

### 3.5. Bernoulli equation for gas

Gas flows are one of the common application areas of Bernoulli’s equation. Gases are compressible fluids but as the flows of incompressible fluids, they are mostly adiabatic and appreciable flow of thermal energy during the flow does not occur.

An ideal gas undergoing adiabatic flow expands while flowing from a region of higher pressure to a lower one and consequently, the work performed by the gas on the surroundings rises lowering the total amount of work done on the fluid. The heat capacity of the gas requires to be defined from its molecular composition to calculate the change in PV for a given change in energy.

To solve the problem it is required to separate the adiabatic, reversible expansion problem from the acceleration problem (Can be compared to solving expanding spring in a gravitational field or the expanding, accelerating spring problem).

\[
W = - \int_0^{V_1} P_1 dV + \int_0^{V_2} P_2 dV = P_1V_1 - P_2V_2 = -\Delta(PV)
\]  (3.5.1)

The equation above is true to calculate the net work done on system by gases flowing horizontally between two regions at fixed pressure.

However the temperature of gas decreases during adiabatic, reversible expansion leading to fall in internal energy and increase in the speed and kinetic energy of the gas flow increases.

For a monatomic ideal gas, the decrease in internal energy amounts to 60% of the increase in kinetic energy. The net change in energy of the gas, which must be equal to the net work done on the gas as it expands, is only 40% of the change in kinetic:

\[
W = \Delta E = \frac{1}{5}Nm\Delta(u)^2
\]  (3.5.2)

For more complex molecules, with \(n\) degrees of freedom contributing to the heat capacity, the change in energy is

\[
\Delta E = \frac{1}{n+2}Nm\Delta(u)^2
\]  (3.5.3)

The internal energy change is negative and smaller than the kinetic energy term. The pressure depends on initial pressure and temperature, \(P_0\) and \(T_0\), and on the ratio of heat capacities, which is (ideally) 7/5 for diatomic molecules and less for polyatomic molecules (Bauman, 1994).

\[
\gamma = \frac{c_p}{c_v}
\]  (3.5.4)

Pressure is equal to multiplication of density, gas constant and absolute temperature of the gas:
Ideal gas with frictionless flow and if there is no heat transfer, then (k=1.4 for air at ambient temperature):

\[ P = \rho \cdot R \cdot T \]  
\[ (3.5.5) \]

\[ \frac{P}{\rho^k} = C(k = \frac{c_p}{c_v}) \]  
\[ (3.5.6) \]

\[ P = C \rho^k \]  
\[ (3.5.7) \]

\[ \frac{2}{1} \frac{dp}{\rho} = \frac{2}{1} \frac{ck \rho^{k-1}d\rho}{\rho} = Ck \frac{2}{1} \rho^{k-2}d\rho = \frac{ck \rho^{k-1}}{k-1} \frac{2}{1} = \frac{k}{k-1} \frac{cp_2^k}{\rho_2} - \frac{cp_1^k}{\rho_1} = \frac{k}{k-1} \frac{p_2}{\rho_2} - \frac{p_1}{\rho_1} \]  
\[ (3.5.8) \]

Then integrated form of the Bernoulli equation is written and substituted with isentropic flow result of ideal gas as below:

\[ g \frac{z_2 - z_1}{\rho} + \frac{2}{1} \frac{dp}{\rho} + \frac{(v_2^2 - v_1^2)}{2} = 0 \]

\[ \frac{2}{1} \frac{dp}{\rho} = \frac{k}{k-1} \frac{p_2}{\rho_2} - \frac{p_1}{\rho_1} \]

\[ g \frac{z_2 - z_1}{\rho} + \frac{k}{k-1} \frac{p_2}{\rho_2} - \frac{p_1}{\rho_1} + \frac{(v_2^2 - v_1^2)}{2} = 0 \]  
\[ (3.5.9) \]

The equation above is Bernoulli equation for ideal gas. As Bernoulli equation has some restrictions for application as the fluid must be steady, frictionless, with no heat transfer etc, the same “musts” are required for gases too (Caretto, 2008).

### 3.6. Practical application examples

Although Bernoulli equation is derived for inviscid flows, it can be applied to real flows to find the values of pressure and velocity at two points along a streamline. Here are some applications of Bernoulli equation.

#### 3.6.1. Pitot Tube

Pitot tube can be used for measuring the velocity of flow, stagnation pressure etc. An example for measuring the velocity of flow in centre point is shown below.

To measure the velocity at the centre of the pipe, point 1 under the piezometer and point 2 at the tip of the Pitot tube are taken and the flow is steady with parallel and straight streamlines. To express the gage pressures at points 1 and 2, the equation is written:

\[ P_1 = \rho g \ h_1 + h_2 \]

\[ P_1 = \rho g \ h_1 + h_2 + h_3 \]  
\[ (3.6.1) \]
After substituting pressures, $V_1$ is calculated (Cengel, 2013):

$$V_1 = \frac{V_1^2}{2g} = \frac{p_2-p_1}{\rho g} = \frac{\rho g (h_1+h_2+h_3-\rho g (h_1+h_2))}{\rho g} = \frac{h_3}{2g}$$

$$V_1 = \frac{2gh_3}{2g} = 2(9.81 \frac{m}{s^2})(0.12 \text{ m}) = 1.53 \text{ m/s}$$

### 3.6.2. Siphon

A siphon aids to draw water from the storage vessel without the use of pump. As shown in figure below, the schematic of siphon can be as simple as hose. If the tank is sufficiently flow, steady flow can be assumed steady. Mass balance of the control volume is

$$\rho V_s A_s = \rho VA$$

Rewriting:

$$V_s = V \frac{A}{A_s}$$

where density is constant; $V_s$ – shows the velocity of free surface falling; $A_s$ – cross sectional area of the free surface less the hose area; $V$ – average velocity in the hose, $A$ – cross sectional area of the hose.

The flowrate out of the hose is:

$$Q = 2A \sqrt{2gH}$$

Bernoulli equation is obtained between point C and the exit-E:

$$\frac{p_c}{\rho} + \frac{1}{2} V_c^2 + gz_c = \frac{p_E}{\rho} + \frac{1}{2} V_E^2 + gz_E$$
Velocities at C and E are same. $z_C = h$ and $z_E = -H$.

![Diagram of Siphon](image1)

**Figure 3.6.2. Siphon (Shaughnessy, 2005)**

### 3.6.3. Prandtl’s static pressure tube

Measure dynamic pressure deviates from that of inviscid flow due to the friction in the fluid. The amount of the deviation depends on the ratio of the diameters and Reynolds number.

![Diagram of Prandtl’s tube](image2)

**Figure 3.6.3. Prandtl’s tube**

By means of Prandtl’s static pressure tube, measurement of the dynamic pressure is determined by combining static pressure tube and the Pitot tube using Bernoulli equation below (Krause, 2005):

$$ p_0 - p_\infty = \beta \frac{\rho}{2} \quad \text{with} \quad \beta = \beta(Re, \frac{d_1}{D}) $$

(3.6.7)
4. Hydraulic losses in pipe flow

Boriss Gjunsburgs, Gints Jaudzems, Roberts Neilands

Riga Technical University, Water Engineering and Technology Department
4.1. Shear stress in laminar flow

The shear stress on the pipe wall can be presented from Newton law and basic equation of hydrodynamics:

\[ \frac{\tau_0}{\gamma} = RL \]

or

\[ \tau_0 = \gamma \cdot R \cdot I \] \hspace{1cm} (4.1.1)

As for pipe hydraulic radius is equal \( \frac{r}{2} \), the equation (4.1.1) can be written as:

\[ \tau = \gamma \cdot \frac{r}{2} \cdot I . \]

The maximum value of \( \tau \) is on the wall of the pipe, when \( r = r_0 \) (Fig. 4.1.2):

\[ \tau_0 = \gamma \cdot \frac{r_0}{2} \cdot I \] \hspace{1cm} (4.1.2)

Dividing \( \tau \) on \( \tau_0 \), the equation for \( \tau \) can be presented:

\[ \tau = \tau_0 \frac{r}{r_0} \] \hspace{1cm} (4.1.3)

According to (4.1.3), the shear stress distribution in pipe at laminar steady flow is showed in Fig. 4.11.

![Fig. 4.1.1. Shear stress distribution at laminar flow](image)

According to Newton law, the shear stress can be found as

\[ \tau = \mu \frac{dV}{dn} \] \hspace{1cm} (4.1.4)

where \( \mu \) – dynamic viscosity; \( \frac{dV}{dn} \) – velocity gradient.

In pressure pipe as a control volume is the cylinder with radius \( r \). The shear stress on the surface pipe wall of it can be determined as:

\[ \tau = \gamma \cdot \frac{r}{2} \cdot I \] \hspace{1cm} (4.1.5)
and according to the Newton law:

$$\tau = \frac{dV}{dn} = -\mu \frac{dV}{dr}$$  \hspace{1cm} (4.1.6)

The values of the shear stress are equal

$$\gamma \cdot \frac{r}{2} \cdot I = -\mu \frac{dV}{dr}$$  \hspace{1cm} (4.1.7)

The velocity can be found as:

$$dV = -\frac{1}{2} \frac{\gamma}{\mu} \cdot I \cdot r \cdot dr$$  \hspace{1cm} (4.1.8)

After integration:

$$V = -\frac{\gamma}{4\mu} \cdot I \cdot r^2 + C$$  \hspace{1cm} (4.1.9)

When $r = r_0$ the velocity is zero: $0 = -\frac{\gamma}{4\mu} \cdot I \cdot r_0^2$ and $C = \frac{\gamma}{4\mu} \cdot I \cdot r_0^2$.

$$V = \frac{\gamma}{4\mu} \cdot I \cdot r_0^2 - r_0^2$$  \hspace{1cm} (4.1.10)

The formula describes parabolic distribution of velocities in cross-section of the pipe in the laminar flow.

At $r = 0$ the velocity is maximal:

$$V_{\text{max}} = \frac{\gamma \cdot I}{4\mu} r_0^2$$  \hspace{1cm} (4.1.11)

The discharge of the flow through elementary cross-section area is

$$dQ = V \cdot d\omega = \nu \cdot 2\pi r dr$$  \hspace{1cm} (4.1.12)

The velocity can be expressed from equation (4.1.10):

$$dQ = \frac{\gamma \cdot I}{4\mu} \cdot \left( r_0^2 - r^2 \right) 2\pi r dr \quad \text{or} \quad dQ = \frac{\gamma \cdot I}{2\mu} \cdot \int_0^r r dr - \int_0^r r^3 dr$$  \hspace{1cm} (4.1.13)

Then the discharge through the cross section area is:

$$Q = \frac{\pi \cdot \gamma \cdot I}{2\mu} \left( r_0^2 - \left( \int_0^r 2 \cdot 0 \cdot 0 + \int_0^r r^3 dr \right) \right) \quad \text{or} \quad Q = \frac{\pi \cdot \gamma \cdot I}{8\mu} r_0^4$$  \hspace{1cm} (4.1.14)
4.2. Shear stress in turbulent flow

In turbulent flow the particles are moving not only straight, but also in transversal direction. The intensive exchange of masses from central part of the pipe to the pipe walls and in opposite direction are taking place.

Intensive mixture of the turbulent flow leads that particles with high mean velocities from central part of the pipe falls to the area near the walls with lower velocities. In that case the velocity of the particle is slowed down. In movement in opposite direction, the particles with low mean velocities reach the area with higher velocities.

In that case low velocity of the particle slows down the movement of the flow.

![Figure 4.2.1. Movement of the particles in turbulent flow](image)

Because of the turbulent mixture of the flow the turbulent shear stress $\tau_2$ are originated. The most accepted approach to find the turbulent shear stress in flow is theory presented by Prandtl. The flow is divided to the laminar sublayer near the pipe walls and turbulent core. In laminar flow the shear stress is raised by viscosity of the liquid, and in turbulent core because of the turbulent mixture.

Then, the shear stress in that model of the flow in pipe is:

$$\tau = \tau_1 + \tau_2$$  \hspace{1cm} (4.2.1)

where $\tau_1$ – shear stress in laminar sublayer; $\tau_2$ – turbulent shear stress.

To find the turbulent shear stress the flow momentum equation is used.

![Figure 4.2.2 Velocity distribution in turbulent flow](image)

Prandtl set the characteristics of the turbulent pulsation of the flow with mean velocity of that flow. Assumptions were made that:

1) Velocity vector of pulsation $V'$, in flow direction is proportional to the mean velocity difference where particle mass were moved by pulsation.
\[ V'_x = -l \frac{dV}{dy} \]  

(4.2.2)

where \( l \) – mixing-length distance; \( dV / dy \) – velocity gradient; \( V \) – temporal mean velocity of turbulent flow, \( y \) – distance from the pipe wall normal to the flow direction.

2) Pulsation part of the velocity \( V'_y \) is proportional to pulsation part of the velocity \( V'_x \) but in opposite direction. To find pulsation shear stress force, the equation of the flow movement is used:

\[ T \cdot dt = \rho \Delta \omega V'_x V'_y \]  

(4.2.3)

or

\[ \frac{T}{\Delta \omega} = \tau = \tau_2 = \rho V'_x V'_y \]  

(4.2.4)

As value of \( V_x \) and \( V_y \) is difficult to find, Prandtl’s mixing length theory in which shear stress in connected with the mean turbulent velocity distribution.

The values of \( V'_x, V'_y \) were presented as \( V'_x = V'_y = l \frac{dV}{dy} \) and then equation is:

\[ \tau_2 = \rho V'_y l \frac{dV_y}{dy} \]  

(4.2.5)

Prandtl supposed that the shear stress at the pipe wall is equal to \( \tau_0 \) and is constant, and the mixing length \( l \) depends on the distance from the pipe wall \( l = k n \)

\[ \tau_2 = \rho l^2 \left( \frac{dV_y}{dn} \right)^2 \quad \text{or} \quad |\tau_2| = \rho l^2 \left( \frac{dV}{dn} \right)^2 \]  

(4.2.6)

where \( l = kn; k \) – universal Prandtl constant \((k = 0.4)\); and \( n \) – distance from the pipe wall.

\[ \frac{\tau_2}{\rho} = \rho k^2 \cdot n^2 \left( \frac{dV}{dn} \right)^2 \]  

(4.2.7)

### 4.2.1. Velocity distribution in turbulent flow

Velocity distribution in turbulent flow can be received from Eq. (4.2.7) the \( dV \) is:

\[ dV = \frac{1}{k} \sqrt{\frac{\tau_0}{\rho}} \frac{dn}{n} \]  

(4.2.8)

where \( \sqrt{\frac{\tau_0}{\rho}} = V_s \) – shear stress velocity; and has dimensions of the velocity.

\[ dV = \frac{V_s}{k} \frac{dn}{n} \]  

(4.2.9)
After integration:

\[ \frac{V}{V_s} = \frac{1}{k} \ln n + C \]  

(4.2.10)

Eq. (4.2.10) is presenting logarithmic law of velocities distribution for turbulent flow. At \( n = r_0 \) the value of velocity is maximal \( V = V_{\text{max}} \), then constant \( C \) is equal to:

\[ C = V_{\text{max}} - \frac{V_s}{k} \ln r_0 \]  

(4.2.11)

The velocity value is:

\[ V = V_{\text{max}} - \frac{V_s}{n} \ln \frac{r_0}{n} \]  

(4.2.12)

### 4.3. Shear stress interaction with hydraulic losses

The pressure flow movement in pipe at laminar flow condition is under consideration and is used to consider shear stress distribution in flow.

The Bernoulli equation for two cross-sections 1-1 and 2-2 (Fig. 4.3.1) can be presented as:

\[ z_1 + \frac{P_1}{\gamma} + \frac{\alpha V_1^2}{2g} = z_2 + \frac{P_2}{\gamma} + \frac{\alpha V_2^2}{2g} + h_i \]  

(4.3.1)

where \( z_1; z_2 \) – geodesic height, \( \frac{P_1}{\gamma}; \frac{P_2}{\gamma} \) – pressure height, \( \frac{\alpha V_1}{2g}; \frac{\alpha V_2}{2g} \) – velocity height and \( h_i \) – hydraulic losses between cross-sections 1-1 and 2-2.

Fig. 4.3.1 Hydraulic losses in pipe at steady flow conditions
For steady flow the mean velocities is not changing along pipe, and then \( \frac{\alpha V_1^2}{2g} = \frac{\alpha V_2^2}{2g} \).

The hydraulic losses for steady flow can be presented from equation (4.3.1) as:

\[
h_i = \left( z_1 + \frac{P_1}{\gamma} \right) - \left( z_2 + \frac{P_2}{\gamma} \right)
\]

The friction force in pipe walls is:

\[
T = \tau_0 \cdot l \cdot \chi
\]

where \( T \) – friction force; \( \tau_0 \) – shear stress on the pipe walls; \( \chi \) – wetted perimeter of the pipe with the round cross-section; \( l \) – distance between two cross-sections.

For steady flow conditions the sum of projection of external forces acting on x-x axis should be equal to zero:

\[
P - T + G \cdot \cos \alpha = 0
\]

where \( P \) – resultant pressure force in sections 1-1 and 2-2; \( T \) – resultant shear stress force; \( G \) – gravity force in distinct part of the pipe line.

![Fig. 4.3.2 Forces in pipe acting in control volume](image)

Then:

\[
\phi_1 - p_2 \omega - \tau_0 \cdot \chi \cdot l + \rho g \omega l \cdot \cos \alpha = 0
\]

where \( p_1 \) and \( p_2 \) – pressure at end cross sections; \( \omega \) – cross-section area; \( \rho \) – fluid density.

As \( l \times \cos \alpha = z_1 - z_2 \), the equation (4.3.4), after dividing on \( \rho g \omega \), can be written as:

\[
\left( z_1 + \frac{P_1}{\gamma} \right) - \left( z_2 + \frac{P_2}{\gamma} \right) = \frac{\tau_0 \cdot \chi \cdot l}{\gamma \cdot \omega} = \frac{T}{\gamma \cdot \omega}
\]

In equation (4.3.6) is presented dependence of linear hydraulic losses from shear stresses on the pipe wall, and as well as, wetted perimeter, length of the pipe line, specific weight, and cross-section area.
\[
\frac{\tau_0 \cdot \chi \cdot l}{\gamma \cdot \omega} = h_i
\]  
(4.3.6)

As \( \frac{\omega}{\chi} = R \), where \( R \) – hydraulic radius; \( \frac{h_i}{l} = I \) – hydraulic slope, \( \omega \) – cross-section area.

Equation (4.3.6) shows the dependence of the hydraulic losses from the shear stress on the pipe wall. With increase of the shear stress value the hydraulic losses is increasing.

### 4.4. Major hydraulic losses

#### Introduction

A water supply pipeline is a system consisting of pipes, pumps, valves and other components. If a transmission system is usually composed of a single-series line than a distribution system consists of a complex network of interconnected pipes. Although flow in pipelines meets continuous changes because of variation of demands, the initial conditions in transient problems are usually assumed to be steady. The design of the steady flow in pipes has hydraulic objective- to secure the desired pressure and flow rate at specific locations in the system, and economic objective to meet the hydraulic requirements with the minimum expense.

The fundamental equation of flow in pipes is Bernoulli’s equation, which is written as relationship between two points and with addition of head loss have a form:

\[
\frac{P_1}{\gamma} + \frac{V_1^2}{2g} + z_1 = \frac{P_2}{\gamma} + \frac{V_2^2}{2g} + z_2 + H_L
\]  
(4.4.1)

where \( \frac{P}{\gamma} \)– pressure head, m; \( V^2/2g \)– velocity head, m; \( z \)– elevation head, m; \( H_L \)– head loss between the two points of the pipeline, m.

The sum of pressure head, velocity head and elevation head can be expressed as total head \( H \) and the Eq.(4.4.1) can be rewritten as

\[
H_1 = H_2 + H_L
\]  
(4.4.2)

The head (or energy) losses result from shear stress along the walls of the pipe, fluid characteristic and from momentum changes at entrances, exits, changes in cross section or direction, and fittings. This Chapter describes head losses in pipes also called as friction losses and main losses. The friction losses are continuous over the length of a pipeline and are a function of the velocity of the flow \( V \), internal diameter of the pipe \( d \), the length of the pipe \( L \), the roughness of pipe’s surface and the fluid characteristic expressed in terms of the kinematic viscosity \( \nu \).

The flow in pipes is expressed by two different flow regimes laminar and turbulent. In laminar flow, streamlines remain parallel to one another and no mixing occurs between adjacent layers. In turbulent flow mixing occurs across the pipe. The flow regime is classified by dimensionless Reynold’s number \( Re \). In pipelines \( Re \) is given as
\[
Re = \frac{VD\rho}{\mu} = \frac{VD}{v}
\]  

(4.4.3)

where \( V \) – the velocity in the pipe; \( D \) – the pipe diameter; \( \rho \) – the density of the fluid; \( \mu \) – the viscosity; \( \nu \) – the kinematic viscosity (\( \mu/\rho \)). The exact value taken to limit the range of \( Re \) vary with author and application. Extensive research on flow regime in pipes with different surface roughness and flow velocity was made by Nikuradse (1933) (Figure 4.4.1).

Figure 4.4.1. Relation of \( \lg(100f) \) to \( \lg(Re) \) according to Nikuradse.
I- Laminar flow; II- turbulent flow in hydraulically smooth pipes; III- turbulent flow in rough pipes. (1- \( R/e = 15 \); 2- \( R/e = 30.6 \); 3- \( R/e = 60 \); 4- \( R/e = 126 \); 5- \( R/e = 252 \); 6- \( R/e = 507 \))

He experimentally measured the resistance to flow posed by various pipes with uniform sand grains with different size \( e \) glued onto the inside walls and found the influence of the relative roughness (ratio of equivalent uniform grain size and the pipe diameter \( e/D \)) on head loss. The relationship between resistance to flow characterized by friction factor \( f \) and Reinold’s number is illustrated by the diagram developed by Moody (1944) (Figure 4.4.2). Commonly the different flow regimes are taken as follows: laminar flow: \( Re \leq 2000 \), transitional flow: \( 2000 < Re \leq 4000 \), turbulent flow: \( Re > 4000 \). These flow regimes have a direct influence on the head loss experienced in a pipeline system.

Figure 4.4.2. Moody diagram illustrating the variation of hydraulic resistance with Reynold’s number.
No matter what flow regime exists within a pipe, a laminar film will exist next to the wall. If the thickness $\delta$ of this film has substantially higher value than the roughness $e$ of the pipe, the pipe will be hydraulically smooth. The actual film thickness can be calculated by formula

$$\delta = \frac{32.8D}{Re \cdot f^{0.5}} \quad (4.4.4)$$

If $e/\delta$ is less than 0.25, the pipe may be considered smooth, while if $e/\delta$ is greater than 6, the pipe is wholly rough. For water mains there is considerable guidance available on the choice of $e$ values to adopt for design. Typical values for new clean pipes and indicative values for design purposes that allow for deterioration of interior conditions are shown in Table 4.4.1.

Table 4.4.1. Values of $e$ for design purposes

<table>
<thead>
<tr>
<th>Description of pipe</th>
<th>$e$, mm</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>New clean pipes</strong></td>
<td></td>
</tr>
<tr>
<td>Steel or ductile iron pipes:</td>
<td></td>
</tr>
<tr>
<td>with spun bitumen or enamel finish</td>
<td>0.025-0.05</td>
</tr>
<tr>
<td>with cement mortar lining</td>
<td>0.03-0.1</td>
</tr>
<tr>
<td>Concrete pipes</td>
<td>0.03-0.3</td>
</tr>
<tr>
<td>Plastic pipes</td>
<td>0.003-0.06</td>
</tr>
<tr>
<td><strong>For design with allowance for deterioration</strong></td>
<td></td>
</tr>
<tr>
<td>Raw water mains</td>
<td>1.5-3.0</td>
</tr>
<tr>
<td>Treated water trunk mains</td>
<td>0.3-1.0</td>
</tr>
<tr>
<td>Distribution systems</td>
<td>0.5-1.5</td>
</tr>
<tr>
<td><strong>For new clean service pipes</strong></td>
<td></td>
</tr>
<tr>
<td>Galvanized steel</td>
<td>0.06-0.30</td>
</tr>
<tr>
<td>Cooper</td>
<td>0.002-0.005</td>
</tr>
<tr>
<td>MDPE</td>
<td>0.003-0.006</td>
</tr>
<tr>
<td>PVC-U</td>
<td>0.003-0.06</td>
</tr>
</tbody>
</table>

4.4.1. **Darcy-Weisbach equation**

A dimensionally correct formula for the head loss is the Darcy-Weisbach equation which gives the head loss in a length of pipe:

$$H_L = f \times \frac{L}{D} \left( \frac{V^2}{2g} \right) \quad (4.4.5)$$

where $f$– friction factor; $L$– pipe length, m; $D$– internal diameter of pipe, m; $V^2/2g$– the velocity head, m.

The diameter of pipe can be found from relation of three variables: $Q$ – discharge, m³/s; $V$ – velocity, m/s; and $A$ – cross section of the flow, m²:

$$Q = AV \quad (4.4.6)$$

For pipelines with circular cross section $A=(\pi D^2)/4$ and diameter can be found by equation:
The friction factor includes the effects of pipe wall roughness and the fluid viscosity, it is not constant but varies with the size of pipe and the degree of turbulence of the flow.

For laminar flow, the friction factor is linearly dependent on the $Re$ with the simple relationship:

$$f = \frac{64}{Re} \quad (4.4.8)$$

For turbulent smooth flow the friction factor is a function of $Re$ (Jeppson, 1976):

$$\frac{1}{\sqrt{f}} = 2 \log \left( e \sqrt{f} \right) \quad (4.4.9)$$

For transition between turbulent smooth and wholly rough flow, the friction factor is a function of both $Re$ and the relative roughness $e/D$. This is most commonly encountered flow regime for water distribution systems and networks. Several equations are used to define friction factor in this flow regime however difference in results is insignificant. Some of more often used are Colebrook-White equation:

$$\frac{1}{\sqrt{f}} = -2 \log \left( \frac{e/D}{3.7} + \frac{2.51}{Re \sqrt{f}} \right) \quad (4.4.10)$$

Swanny and Jain (1976) equation which is included in the popular network-analysis program EPANET and several other codes:

$$f = \frac{0.25 \left[ \log \left( \frac{e}{3.7D} + \frac{5.74}{Re^{0.89}} \right) \right]^2}{\log \left( \frac{e}{3.7D} + \frac{5.74}{Re^{0.89}} \right)} \quad (4.4.11)$$

or Altshul (1975) equation:

$$f = 0.11 \left( \frac{68}{Re} + \frac{e}{D} \right)^{0.25} \quad (4.4.12)$$

When the flow described as “completely” turbulent, $f$ becomes independent of $Re$ and depends only on relative roughness. Friction factor in this regime can be found by Nikuradse equation:

$$\frac{1}{\sqrt{f}} = 1.14 - 2 \log \left( e/D \right) \quad (4.4.13)$$

### 4.4.2. Hazen-Williams equation

Beside to the Darcy-Weisbach equation, a number of empirical equations which are more easily solved have been developed. The Hazen-Williams equation is such a relation and has a form:
\[ H_L = \frac{10.67 \cdot L \cdot Q^{1.85}}{C^{1.85} \cdot D^{4.87}} \]  

(4.4.14)

where \( L \) – pipe length, m; \( Q \) – discharge, \( \text{m}^3/\text{s} \); \( D \) – pipe inside diameter, m; \( C \) represents an equivalent friction factor (150 for smooth and 75 for rough, dirty pipe surface). It is modified by \( Re \) and the equation is therefore not totally divorced from the actual phenomena which occur in pipeline flow. Note that number 10.67 is a factor for SI units only (4.52 for US units). Accepted values of \( C \) for pipes of different materials are presented in Table 4.4.2.

Table 4.4.2. Hazen-Williams coefficient \( C \) for various pipe materials

<table>
<thead>
<tr>
<th>Description of pipe</th>
<th>Value of ( C )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cast iron:</td>
<td></td>
</tr>
<tr>
<td>New</td>
<td>130</td>
</tr>
<tr>
<td>5 years old</td>
<td>120</td>
</tr>
<tr>
<td>10 years old</td>
<td>110</td>
</tr>
<tr>
<td>20 years old</td>
<td>90-110</td>
</tr>
<tr>
<td>30 years old</td>
<td>75-90</td>
</tr>
<tr>
<td>Concrete</td>
<td>120</td>
</tr>
<tr>
<td>Cement lined</td>
<td>140</td>
</tr>
<tr>
<td>Welded steel</td>
<td>120</td>
</tr>
<tr>
<td>Riveted steel</td>
<td>110</td>
</tr>
<tr>
<td>Plastic</td>
<td>150</td>
</tr>
<tr>
<td>Asbestos cement</td>
<td>140</td>
</tr>
</tbody>
</table>

**General form of equations**

Pipe flow \( Q \) and energy loss caused by friction \( H_L \) in individual pipes can be represented by a number of equations and general relationship of them has the form:

\[ H_L = K Q^n \]  

(4.4.15)

where \( K \)– pipe coefficient that depends on pipe’s diameter, length and surface roughness and \( n \) is an exponent in a range of 2. For Darcy-Weisbach equation \( n=2 \):

\[ H_L = \left[ \frac{8 \cdot f \cdot L}{g \cdot \pi^2 \cdot D^5} \right] Q^2 = K Q^2 \]  

(4.4.16)

For Hazen-Williams equation \( n = 1.85 \):

\[ H_L = \left[ \frac{10.67 \cdot L}{C^{1.85} \cdot D^{4.87}} \right] Q^{1.85} = K Q^{1.85} \]  

(4.4.17)

It should be noted that calculation of pipe coefficient \( K \) differs according to the equation used. \( K \) is a constant in turbulent flow that is commonly assumed to occur in the water distribution systems.
4.4.3. **Hydraulically equivalent pipes**

Multiple pipe systems have series and parallel configuration. The analysis of such systems becomes easier if equivalent single pipe can be found that will have same relationship between head loss and flow rate as the original complex configuration.

**Pipes configured in series**

Pipes configured in series (Figure 4.4.3) can vary in pipe size, type and length however discharge through each pipe is the same.

![Figure 4.4.3. Pipes configured in series](image)

The total head loss consists of the sum of head loss in each separate segment, and using common equation (4.4.15):

\[
H_L = \sum_{i \in I_p} H_{L,i} = \sum_{i \in I_p} K_i Q^n = K_e Q^n
\]

(4.4.18)

where \( I_p \) – set of pipes in the series of pipes; \( K_e \) – pipe coefficient for the equivalent pipe:

\[
K_e = K_1 + K_2 + K_3 + \ldots = \sum_{i \in I_p} K_i
\]

(4.4.19)

As coefficient \( K_e \) is independent of the flow rate, assumption of \( Q \) is not required.

**Pipes configured in parallel**

Pipes configured in parallel (Figure 4.4.4) can vary in size, flow rate, length and pipe material however head loss in each pipe must be the same because they have common end points, or

\[
H_A - H_B = H_{L,1} = H_{L,2} = H_{L,3}
\]

(4.4.20)

where \( H_A \) and \( H_B \) – pressure head at points A and B, respectively, m; and \( H_{L,1}, H_{L,2}, H_{L,3} \) – head loss in parallel pipes 1, 2 and 3, respectively, m.
The flow rate in the upstream and downstream pipes must be equal the sum of the flow in parallel pipes, because of the mass conservation law:

\[ Q = Q_1 + Q_2 + Q_3 = \sum_{m \in M_P} Q_m \]  

(4.4.21)

where pipe \( m \) is in the set of parallel pipes \( M_P \). Using Eq.(4.4.15) flow rate \( Q = (H_L/K)^{1/n} \). Substituting this in Eq. (4.4.21) and taking into account that head loss for each pipe is similar (Eq.4.4.20):

\[ Q = H_L^{\frac{1}{n}} \left[ \left( \frac{1}{K_1} \right)^{\frac{1}{n}} + \left( \frac{1}{K_2} \right)^{\frac{1}{n}} + \left( \frac{1}{K_3} \right)^{\frac{1}{n}} + \ldots \right] = H_L^{\frac{1}{n}} \sum_{m \in M_P} \left( \frac{1}{K_i} \right)^{\frac{1}{n}} = H_L^{\frac{1}{n}} \left( \frac{1}{K_e} \right)^{\frac{1}{n}} \]  

(4.4.22)

Dividing parts of Eq. (4.4.22) by \( H_L^{\frac{1}{n}} \):

\[ \sum_{m \in M_P} \left( \frac{1}{K_i} \right)^{\frac{1}{n}} = \left( \frac{1}{K_e} \right)^{\frac{1}{n}} \]  

(4.4.23)

\( K_e \) can be found after calculation of \( K \) for each parallel pipe. Head loss across the parallel pipes can be found by substituting of \( K_e \) to Eq. (4.4.15).

The flow in each pipe can be calculated using found \( H_L \) and \( K \) of each parallel pipe by common Eq. (4.4.15).

### 4.4.4. Pipe networks

The water distribution system in urban area is complex because of large number of pipelines and their differences in sizes, length and type. Analysis of pipe networks containing loops is complicate because of interaction of each separate part of it. The laws of conservation of mass and energy must be satisfied for flow in pressurized pipe systems under steady conditions. The law of conservation of mass define that the rate of storage in the system is equal to the difference between the inflow and the outflow of the system:

\[ \sum Q_{in} - \sum Q_{out} = q_{ext} \]  

(4.4.24)
where $Q_{in}$ and $Q_{out}$ – pipe flow rates into and out of the node, respectively; and $q_{ext}$ – external demand or supply.

The law of conservation of energy states that the difference energy between two points is equal to the frictional and minor losses and the energy added to the flow components between these points:

$$\sum_{i \in l_p} H_{L,i} + \sum_{j \in l_p} H_{P,j} = \Delta E$$  \hspace{1cm} (4.4.25)

where $H_{L,i}$ – head loss across component $i$ along the path; $H_{P,j}$ – the head added by pump $j$; and $\Delta E$ – difference in energy between the end point of the path. Signs are applied to each term in Eq. (4.4.25) according to flow direction relative to moving clockwise around the loop. Energy loss for pipes with flow in the clockwise direction would be positive, and those in the counter-clockwise directions are given a negative sign. For added energy by pumps there is a negative sign for clockwise direction and positive sign for opposite direction.

**Hardy Cross method**

The Hardy Cross method was developed in 1936 by Cross and is frequently used to solve simply loop systems by hand. For large systems computer programs are used.

![Figure 4.4.5. Example network](image)

The pipe network on Fig.4.4.5 includes three loops marked by roman numerals, eight nodes marked by capital letters, ten pipes marked by Arabic numerals and each pipe has specific flow rate. The Loop I includes Pipe 1, Pipe 2, Pipe 3 and Pipe 4, the Loop II includes Pipe 3, Pipe 5, Pipe 6 and Pipe 7. Note that the Pipe 3 belongs to the both, Loop I and Loop II. Each node must satisfy the law of conservation of mass, and each loop must satisfy the law of conservation of energy. According to that the Loop I incorporate following equations:
\[
\sum_{A} Q = Q_{A} - Q_{1} - Q_{2} = 0 \\
\sum_{B} Q = Q_{2} - Q_{5} - Q_{3} = 0 \\
\sum_{D} Q = Q_{1} - Q_{D} + Q_{4} = 0 \\
\sum_{E} Q = Q_{3} + Q_{7} - Q_{4} + Q_{8} = 0 \\
\sum_{L} H_{L} = -K_{1}Q_{1}^n + K_{2}Q_{2}^n + K_{3}Q_{3}^n + K_{4}Q_{4}^n = 0 
\]

It must be taken into account that flow direction in pipes can change. Diameter, type and length of pipes are known also elevations of nodes are known. The unknown parameter \( Q_n \), where \( n \) is numbers from 1 to 10 must be found; also corresponding flow direction in each pipe must be defined. The problem will be solved when all equations of the Loops I, II and III will be satisfied and it can be done using trial and error method. The most known and simply is mentioned before Hardy Cross method. The following terms must be noted using this method:

- solution of the problem starts by assuming well-founded initial flow rates and flow directions for each pipe in the pipe network;
- calculation of each pipe should be made by iteration method while the law of conservation of energy is fulfilled for all of the loops in pipe network; for calculation by hand following discrepancy is allowed in results of calculated head loss: ±0.3-0.5m for separate loops and ±1.0-1.5m for contour-loop of the pipe network;
- the more accurately initial flow rates for pipes are assumed, the less inadequacy of loops flow rate \( \Delta Q \) will be found on each step of calculation;
- if flow rate inadequacy of the loop is found, it must be used for each pipe in the loop to correct assumed flow. If pipe belongs to one only loop than flow correction equation is:

\[
Q_n = Q_{0n} \pm \Delta Q 
\]

(4.4.26)

where \( Q_{0n} \) is initially assumed or calculated flow rate and \( \Delta Q \) is flow correction value of the corresponding loop. The sign “+” or “-” depends on the flow direction in pipe and on the sign of calculated \( \Delta Q \). For the Loop I of the pipe network showed on Fig.4.4.5 the flow correction equations for each pipe are following:

\[
Q_1 = Q_{01} + \Delta Q_1, \quad Q_2 = Q_{02} + \Delta Q_2, \quad Q_4 = Q_{04} + \Delta Q_4, \quad Q_3 = Q_{03} + \Delta Q_3 - \Delta Q_{II} 
\]

Since the Pipe 3 belongs to the both Loop I and Loop II, the flow rate of this pipe must be corrected by the flow correction values of both loops.

In general for the head loss for each loop can be found using Eq. (4.4.15):

\[
\sum_{L} H_{L} = \sum_{L} K_{L}Q_{L}^n = 0 
\]

(4.4.27)

The flow correction value for each loop is found by equation
\[ \Delta Q = \frac{\sum L K_i Q_{0i}^n}{\sum L nK_i Q_{0i}^{n-1}} \]  

(4.4.28)

If Darcy-Weisbach equation is used for this method, than \( K_i \) is found according to Eq.(4.4.16) and the flow correction value by equation

\[ \Delta Q = -\frac{\sum L K_i Q_{0i}^2}{\sum L 2K_i Q_{0i}^1} \]  

(4.4.29)

4.4.5. Major head loss in gas pipelines

Transporting of piped gas (air, steam, natural and artificial gas) are commonly used for industrial and for domestic purposes. Gases are classified as compressible fluids because unlike liquids, gases are subject to large variations in volume with changes in pressure and temperature. Frictional pressure loss of the gas pipelines will be illustrated with examples in this chapter.

The flowing gas in comparison with flowing liquid is characterised by difference in physical properties of gaseous fluids. In hydraulic calculation of gas pipelines two cases should be considered: flow by small relative pressure difference and flow by high relative pressure difference. It is described by relation of absolute difference of pressure between two points of pipeline \( \Delta P \) and average pressure between those points \( P_A = (P_1 + P_2)/2 \):

\[ \Delta P / P_A \]  

(4.4.30)

The fluid compressibility can by neglected if \( \Delta P / P_A \) < 5% and density of the fluid can be assumed invariable in the length of the pipeline. Calculation of friction loss for such gas pipelines can be held in same way as for hydraulic calculation for flowing liquid, described previously.

In case of high relative pressure difference, when \( \Delta P / P_A > 5\% \), neglecting of compressibility is not allowed, and pressure drop of transporting fluid in whole pipeline length must be considered. It is also connected with reduction of the gas density and increasing of the velocity in the direction of flow.

Bernoulli’s equation for gas flow in pipelines has form:

\[ \rho \phi (z_1 - z_2) = \frac{\rho (V_1^2 - V_2^2)}{2} + z = H_L \]  

(4.4.31)

where \( H_L \) describes pressure loss in N/m².

In most of practical cases component \( \rho g(z_1 - z_2) \) can be neglected because density of gas has relatively low value comparing to other components of this equation. Therefore it can be rewritten in the form

\[ p_1 + \frac{\rho V_1^2}{2} = p_2 + \frac{\rho V_2^2}{2} + H_L \]  

(4.4.32)
where \( \rho v^2/2 \) is the dynamic pressure.

Than equation for main head loss calculation in gas pipelines has a form

\[
H_L = f \frac{L}{D} \frac{V^2}{2},
\]

(4.4.33)

and equation to determine minor head loss:

\[
h_m = K \frac{V^2}{2}
\]

(4.4.34)

where \( h_m \) – minor losses, m; \( K \) – minor loss coefficient.

4.4.5.1. Hydraulic calculation of a low-pressure gas pipelines

The value of local pressure drops is small compared to the frictional pressure losses in long pipelines of gas flow, so it can be assumed that the total pressure drop is approximately equal to the linear pressure losses and Eq. (4.4.33) can be used to determine the overall pressure loss over the entire length of the gas pipeline.

Substituting in Eq. (4.4.33), the value \( f \) of Eq. (4.4.12), the common formula to determine pressure loss in gas pipeline takes the form:

\[
H_L = 7 \left( \frac{e}{D} + \frac{1922DV}{Q} \right) \frac{\gamma Q^2}{D^5}
\]

(4.4.35)

where \( H_{L,i} \) – the head loss, mm water column; \( e \) – the equivalent uniform grain size, cm; \( D \) – the internal diameter of pipe, m; \( v \) – the kinematic coefficient, m²/s; \( \gamma \) – specific weight of gas, kgs/m³; \( L \) – the pipe length, m.

4.4.5.2. Hydraulic calculation of gas pipelines at high differential pressures

For the long gas pipelines with high difference of pressure at the start and the end points of pipeline, the difference of gas density and the reduction of gas pressure along the length of the pipeline must be taken into account during head loss calculation. In other words dependences of density, velocity and friction factor \( \rho = f(L); v = f_1(L); f = f_2(L) \) must be known. The friction factor \( f \) is a function of Reynold’s number \( Re \), equivalent uniform grain size \( e \) and the internal diameter of pipe \( D \). The equivalent uniform grain size doesn’t change along the length of pipeline and Reynold’s number can be submitted in the form:

\[
Re = \frac{\rho V D}{\mu} = \frac{4 \rho Q}{\pi D \mu} = \frac{4 M}{\pi D \mu}
\]

(4.4.36)

where \( \mu \) – a gas viscosity; \( M \) – a mass flux or mass flow rate.

In isothermal regime dynamic viscosity does not change along the length of the pipeline and it means Reynold’s number and finally the friction factor also remains constant.

In most practical cases, an isothermal gas flow in the turbulent regime occurs. For such regime calculation of pressure loss can be made by formula
\[
\frac{P_1^2 - P_2^2}{L} = 1.45 \left( \frac{e}{D} + \frac{1922D\nu}{Q} \right)^{0.25} \frac{Q^2}{D^3}
\]

(4.4.37)

where \(P_1\) and \(P_2\) – absolute gas pressure at the start and the end point of pipeline respectively, \(\nu\); \(L\) – a length of pipeline, km; \(D\) – the internal diameter of pipe, m; \(e\) – the equivalent uniform grain size, cm; \(\gamma\) – specific weight of gas kgs/m\(^3\); \(Q\) – a gas flow, m\(^3\)/h; \(\nu\) – the kinematic coefficient, m\(^2\)/s.

The values \(\gamma\), \(Q\) and \(\nu\) are given at normal conditions (temperature 0°C and pressure 0.1MPa). Eq. (4.4.37) presents common formula valid in the entire region of turbulent regime.

For special cases Eq. (4.4.37) must be used in different form. When \(e/D\) is considerably less than \(1922D\nu/Q\):

\[
\frac{P_1^2 - P_2^2}{L} = 9.6 \frac{\nu^{0.25} \gamma^{0.175}}{D^{4.75}}
\]

(4.4.38)

in turn if \(e/D\) is considerably higher than \(1922D\nu/Q\):

\[
\frac{P_1^2 - P_2^2}{L} = 1.45 \frac{Q^2 \gamma e^{0.25}}{D^{3.28}}
\]

(4.4.39)

Eq. (4.4.39) is valid and can be applied at high gas velocities (\(V > 50\) m/s).

4.4.5.3. Head loss in ventilation ducts

Ventilation ducts usually have rectangular or square cross sections therefore equivalent diameter \(D_e\) in Eq. (4.4.31) must be used:

\[
H_L = f \frac{L}{D_e} \rho \frac{V^2}{2}
\]

(4.4.40)

And friction factor \(f\) can be calculated by Eq. (4.4.12).

Major head loss for ventilation ducts mostly have less values than minor head loss therefore it is important to calculate minor head loss accurately.

4.4.6. Practical application examples

**Example 1.** A plastic pipe with internal diameter 176.2mm carries a flow of 0.05m\(^3\)/s. The length of pipe is 250m. What is the friction loss in the pipe? Roughness of pipe \(e = 0.01\), water \(T = 20°C\).

**Solution:** According to Eq.4.4.7 flow velocity can be found:

\[
V = \frac{4Q}{\pi D^2} = \frac{4 \cdot 0.05}{\pi \cdot 0.1762^2} = 2.05 \text{ m/s}
\]

The Reynold’s number now can be found taking into account kinematic viscosity for water at \(T=20°C\) is \(1.004 \times 10^{-6}\) m\(^2\)/s:
Re = \frac{VD}{\nu} = \frac{2.05 \cdot 0.1762}{1.004 \cdot 10^{-6}} = 359771

As it is turbulent flow, the friction factor can be found by Altshul (Eq.4.4.12):

\[ f = 0.11 \left( \frac{68}{359771} + \frac{0.01}{176.2} \right)^{0.25} = 0.11 \cdot 0.1252 = 0.0138 \]

And friction loss by Darcy-Weisbach (Eq.4.4.5):

\[ H_L = 0.0138 \frac{250}{0.1762} \left( \frac{2.05^2}{2 \cdot 9.806} \right) = 0.0138 \cdot 1418.84 \cdot 0.2143 = 4.2 \text{ m} \]

**Example 2.** The welded steel pipe has an inside diameter of 317.5mm. How much flow can this pipe carry if the allowable head loss is 6.3m in a length of 300m?

**Solution:** from Table 4.4.2 the Hazen-Williams coefficient C for pipe is 120. All values for Eq.4.4.14 are known excepting discharge \( Q \). After modifying of Hazen-Williams equation:

\[ Q = \left( \frac{H_L \cdot C^{4.85} \cdot D^{4.85}}{10.67 L} \right)^{1.85} = \left( \frac{6.3 \cdot 120^{1.85} \cdot 0.3175^{4.85}}{10.67 \cdot 300} \right)^{1.85} = 0.204 \text{ m}^3 / s \]

**Example 3.** For the three pipes in series in Fig. 4.4.3 find the equivalent pipe coefficient, calculate the discharge in the pipes if the total head loss is 3.8m. Determine the piezometric head at points b, c, and d if the total energy at the inlet (point a) is 30m.

**Solution:** substituting the appropriate values, the pipe coefficient \( K \) for the Hazen-Williams equation for each pipe (Eq. 4.4.17):

\[ K_1 = \frac{10.67 \cdot 100}{120^{1.85} \cdot 0.254^{4.87}} = 168.51; \quad K_2 = \frac{10.67 \cdot 200}{120^{1.85} \cdot 0.203^{4.87}} = 716.48; \]

\[ K_3 = \frac{10.67 \cdot 50}{80^{1.85} \cdot 0.152^{4.87}} = 1551.81 \]

The equivalent \( K_e \) is the sum of individual pipes (Eq. 4.4.10), or \( K_e = 2436.80 \)

Using the equivalent head loss coefficient \( K_e \) and Eq.4.4.15, the flow rate can be found:

\[ Q = \left( \frac{3.8}{2436.8} \right)^{1.85} = 0.0304 \text{ m}^3 / s \]

Now this relationship and \( K_e \) can be used for any flow rate and head loss. Thus, for example, if the flow rate will be 0.05 m$^3$/s, the head loss by Eq.4.4.17 will be \( H_L = 2436.8 \cdot 0.05^{1.85} = 9.55 \) m. The single pipe with head loss coefficient \( K_e = 2436.8 \) is equivalent to the three pipe series in Fig. 4.4.3.

The total energy at point b is the total energy at the source minus the head loss in the first pipe segment:

\[ H_b = H_a - H_{L,1} = 30 - K_1 Q^{1.85} = 30 - 168.51 \cdot 0.0304 \cdot 0.85 = 29.74 \text{ m} \]

Similarly, the head losses in the second and third pipes are 1.12m and 2.42m, respectively. Thus, the piezometric head at the point c and d are 28.62m and 26.2m, respectively.

**Example 4.** Determine the head loss between points A and B for three parallel pipes (Fig. 4.4.4). The total system flow is 0.2m$^3$/s. Find the flow in each pipe.
Solution. The head loss coefficient $K$ for each pipe must be determined (Eq. 4.4.17):

$$K_1 = \frac{10.67 \cdot 120}{100^{1.85} \cdot 0.25^{4.87}} = 218.46; K_2 = \frac{10.67 \cdot 100}{80^{1.85} \cdot 0.4^{4.87}} = 27.89; K_3 = \frac{10.67 \cdot 150}{120^{1.85} \cdot 0.3^{4.87}} = 80.2$$

The equivalent $e$ is found from Eq. (4.4.23):

$$\left( \frac{1}{K_1} \right)^{1.85} + \left( \frac{1}{K_2} \right)^{1.85} + \left( \frac{1}{K_3} \right)^{1.85} = \left( \frac{1}{K_e} \right)^{1.85} = 0.3133, \text{ and } K_e = 8.56.$$  

By Eq. (4.4.17) the head loss $H_L = 8.56 \cdot 0.2^{1.85} = 0.436 \text{ m}.$

According to Eq. (4.4.20) head loss in each parallel pipe is equal to that between points A and B. The flow in each pipe can be calculated using the individual pipe’s coefficient $K$ and Eq. (4.4.17):

$$Q_1 = \left( \frac{0.436}{218.46} \right)^{1.85} = 0.0347 \text{ m}^3 / \text{s}; Q_2 = \left( \frac{0.436}{27.89} \right)^{1.85} = 0.1056 \text{ m}^3 / \text{s};$$

$$Q_3 = \left( \frac{0.463}{80.2} \right)^{1.85} = 0.0597 \text{ m}^3 / \text{s}$$

The sum of discharge in each pipe must be verified regarding to system flow $Q = 0.2 \text{ m}^3 / \text{s}: Q = Q_1 + Q_2 + Q_3 = 0.0347 + 0.1056 + 0.0597 = 0.2 \text{ m}^3 / \text{s}.$

Example 5. Find out outflow $Q_F$ of the node F; determine flow rate and flow direction for pipe network shown in Fig. 4.4.6. Inside diameters for pipes are given. Pipe surface roughness is 0.10 and is equal for all pipes. The kinematic viscosity of fluid (water $T=10°C$) $\nu = 0.013 \text{ St} = 1.3 \cdot 10^{-6} \text{ m}^2 / \text{s}.$

![Figure 4.4.6. Pipe network example](image)

**Solution:** According to the law of mass conservation all incoming and outgoing flows must equal to 0. Flow rate $Q_F$ of the node F can be found:

$$Q_F = Q_A - Q_C - Q_B = 1.6 - 0.25 - 0.40 = 0.95 \text{ m}^3 / \text{s}$$

Initial flow directions and flow rates for all pipes of the network must be assumed. Assumption for first calculation step is shown on Fig. 4.4.7
Mass conservation law for each node should be checked after first assumption and after each further flow corrections in pipes. Solution of this problem will be made using Darcy-Weisbach method, so Eq. (4.4.16) and Eq. (4.4.29) will be applied. The friction factor \( f \) will be calculated by Altshule (Eq. 4.4.12), flow velocity \( V \) by Eq. (4.4.7) and Reynold’s number \( Re \) by Eq. (4.4.3). Convenient way for calculation is combining of all calculation data to the table (Table 4.4.3) including pipe numbers, initial data (\( L \) and \( D \)) and calculated data (\( V \), \( Re \), \( f \), \( K \), \( H_L \) and \( 2KQ \)). Column indicating the sign according to the flow direction in the loop should be included as well.

Table 4.4.3. Calculation data table for solution of the pipe network

<table>
<thead>
<tr>
<th>Pipe No</th>
<th>L (m)</th>
<th>D (m)</th>
<th>( Q_{in} ) (m(^3)/s)</th>
<th>+ / -</th>
<th>V (m/s)</th>
<th>( Re )</th>
<th>f</th>
<th>K</th>
<th>( H_L ) m</th>
<th>2KQ</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>300</td>
<td>0.50</td>
<td>0.20</td>
<td>-</td>
<td>1.02</td>
<td>392308</td>
<td>0.015</td>
<td>11.90</td>
<td>0.48</td>
<td>4.76</td>
</tr>
<tr>
<td>2</td>
<td>500</td>
<td>0.75</td>
<td>1.40</td>
<td>+</td>
<td>3.17</td>
<td>1828846</td>
<td>0.013</td>
<td>2.26</td>
<td>4.43</td>
<td>6.33</td>
</tr>
<tr>
<td>3</td>
<td>400</td>
<td>0.35</td>
<td>0.10</td>
<td>+ / -</td>
<td>1.04</td>
<td>280000</td>
<td>0.017</td>
<td>106.98</td>
<td>1.07</td>
<td>21.40</td>
</tr>
<tr>
<td>4</td>
<td>500</td>
<td>0.50</td>
<td>0.20</td>
<td>+</td>
<td>1.02</td>
<td>392308</td>
<td>0.015</td>
<td>19.83</td>
<td>0.79</td>
<td>4.48</td>
</tr>
<tr>
<td>5</td>
<td>500</td>
<td>0.50</td>
<td>1.30</td>
<td>+</td>
<td>6.62</td>
<td>2546154</td>
<td>0.013</td>
<td>17.19</td>
<td>29.05</td>
<td>44.69</td>
</tr>
<tr>
<td>6</td>
<td>300</td>
<td>0.25</td>
<td>1.05</td>
<td>+</td>
<td>21.59</td>
<td>4113462</td>
<td>0.016</td>
<td>406.13</td>
<td>447.76</td>
<td>852.87</td>
</tr>
<tr>
<td>7</td>
<td>500</td>
<td>0.40</td>
<td>0.10</td>
<td>+</td>
<td>0.80</td>
<td>246154</td>
<td>0.017</td>
<td>68.59</td>
<td>0.69</td>
<td>13.72</td>
</tr>
</tbody>
</table>

Now the total head loss for Loop I and Loop II can be found. The sign of pipe should be taken into account during this calculation:

\[
\sum_{L_I} H_L = -H_{L1} + H_{L2} + H_{L3} + H_{L4} = -0.48 + 4.43 + 1.07 + 0.79 = 5.81m
\]

\[
\sum_{L_{II}} H_L = -H_{L3} + H_{L5} + H_{L6} + H_{L7} = -1.07 + 29.05 + 447.76 + 0.69 = 476.43m
\]

The sum of values 2KQ for each loop. Signs are avoided:

\[
\sum_{L_I} 2KQ = 4.76 + 6.33 + 21.4 + 7.93 = 40.42
\]

\[
\sum_{L_{II}} 2KQ = 21.4 + 44.69 + 852.87 + 13.72 = 932.68
\]

and correction flow \( \Delta Q \) for each loop:

\[
\Delta Q_{L_I} = \frac{-5.81}{40.42} = -0.144 \text{ m}^3 / \text{s}
\]

\[
\Delta Q_{L_{II}} = \frac{-476.43}{932.68} = -0.511 \text{ m}^3 / \text{s}
\]
Now the flow correction for each pipe of the network must be made by adding correction flow. The sign of pipe should be taken into account during this calculation

\[ Q_{11} = -Q_{01} + \Delta Q_{LI} = -0.20 - 0.144 = -0.344 \, m^3/s \]
\[ Q_{12} = Q_{02} + \Delta Q_{LI} = 1.4 - 0.144 = 1.256 \, m^3/s \]
\[ Q_{13} = Q_{03} + \Delta Q_{LI} - \Delta Q_{LH} = 0.10 - 0.144 + 0.511 = 0.467 \, m^3/s \]
\[ Q_{14} = Q_{04} + \Delta Q_{LI} = 0.20 - 0.144 = 0.056 \, m^3/s \]
\[ Q_{15} = Q_{05} + \Delta Q_{LH} = 1.3 - 0.511 = 0.789 \, m^3/s \]
\[ Q_{16} = Q_{06} + \Delta Q_{LH} = 1.05 - 0.511 = 0.539 \, m^3/s \]
\[ Q_{17} = Q_{07} + \Delta Q_{LH} = 0.1 - 0.511 = -0.411 \, m^3/s \]

The sign achieved during this calculation must be compared with the sign of pipeline (flow direction) assumed before. If the sign changes also flow direction must be changed. The “+” sign changed to the “−” sign for the pipe No7 during this calculation, therefore flow direction for this pipe must be turned to opposite side. After adding of new flow rates to the network scheme, confirmation to mass conservation law must be checked. The results of flow rates and directions corrected in accordance to first calculation are shown on Fig. 4.4.8.

![Diagram showing flow rates and direction corrections](image)

**Figure 4.4.8.** Intermediate results after first calculation step

The results of second-step calculation presented in the Table 4.4.4.

<table>
<thead>
<tr>
<th>Pipe No</th>
<th>L (m)</th>
<th>D (m)</th>
<th>( Q_{18} ) (m(^3)/s)</th>
<th>( +/− )</th>
<th>( V ) (m/s)</th>
<th>( Re )</th>
<th>( f )</th>
<th>( K )</th>
<th>( H_L ) (m)</th>
<th>2KQ</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>300</td>
<td>0.50</td>
<td>0.344</td>
<td>-</td>
<td>1.75</td>
<td>673077</td>
<td>0.014</td>
<td>11.11</td>
<td>1.31</td>
<td>7.64</td>
</tr>
<tr>
<td>2</td>
<td>500</td>
<td>0.75</td>
<td>1.256</td>
<td>+</td>
<td>2.84</td>
<td>1638462</td>
<td>0.013</td>
<td>2.26</td>
<td>3.57</td>
<td>5.68</td>
</tr>
<tr>
<td>3</td>
<td>400</td>
<td>0.35</td>
<td>0.467</td>
<td>+/−</td>
<td>4.85</td>
<td>1305769</td>
<td>0.015</td>
<td>94.39</td>
<td>20.59</td>
<td>88.16</td>
</tr>
<tr>
<td>4</td>
<td>500</td>
<td>0.50</td>
<td>0.056</td>
<td>+</td>
<td>0.29</td>
<td>111538</td>
<td>0.019</td>
<td>25.12</td>
<td>0.08</td>
<td>2.81</td>
</tr>
<tr>
<td>5</td>
<td>500</td>
<td>0.50</td>
<td>0.789</td>
<td>+</td>
<td>4.02</td>
<td>1546154</td>
<td>0.014</td>
<td>18.51</td>
<td>11.52</td>
<td>29.21</td>
</tr>
<tr>
<td>6</td>
<td>300</td>
<td>0.25</td>
<td>0.539</td>
<td>+</td>
<td>10.98</td>
<td>2111538</td>
<td>0.016</td>
<td>406.13</td>
<td>117.99</td>
<td>437.81</td>
</tr>
<tr>
<td>7</td>
<td>500</td>
<td>0.40</td>
<td>0.411</td>
<td>-</td>
<td>3.27</td>
<td>1006154</td>
<td>0.015</td>
<td>60.52</td>
<td>10.22</td>
<td>49.75</td>
</tr>
</tbody>
</table>

The flow correction values found during this calculation:
\[
\Delta Q_{L1} = \frac{22.93}{104.29} = -0.22 \text{ m}^3/\text{s}
\]
\[
\Delta Q_{LII} = \frac{-98.7}{604.93} = -0.163 \text{ m}^3/\text{s}
\]

The correction flow values of following calculation steps are shown in Table 4.4.5:

Table 4.4.5. Results of calculation steps 2 to 5

<table>
<thead>
<tr>
<th></th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
</tr>
</thead>
<tbody>
<tr>
<td>ΔQ_{L1} (m³/s)</td>
<td>-0.14</td>
<td>-0.053</td>
<td>-0.021</td>
<td>-0.006</td>
</tr>
<tr>
<td>ΔQ_{LII} (m³/s)</td>
<td>-0.064</td>
<td>-0.029</td>
<td>-0.009</td>
<td>-0.004</td>
</tr>
</tbody>
</table>

The sixth calculation step gives following results shown in Table 4.4.6:

Table 4.4.6. Results of sixth calculation step

<table>
<thead>
<tr>
<th>Pipe No</th>
<th>L (m)</th>
<th>D (m)</th>
<th>(Q_{in}) (m³/s)</th>
<th>+ / -</th>
<th>V (m/s)</th>
<th>(Re)</th>
<th>(f)</th>
<th>(K)</th>
<th>(H_L) (m)</th>
<th>2KQ</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>300</td>
<td>0.50</td>
<td>0.784</td>
<td>-</td>
<td>3.99</td>
<td>1534615</td>
<td>0.014</td>
<td>11.11</td>
<td>6.83</td>
<td>17.42</td>
</tr>
<tr>
<td>2</td>
<td>500</td>
<td>0.75</td>
<td>0.816</td>
<td>+</td>
<td>1.85</td>
<td>1067308</td>
<td>0.013</td>
<td>2.26</td>
<td>1.5</td>
<td>3.69</td>
</tr>
<tr>
<td>3</td>
<td>400</td>
<td>0.35</td>
<td>0.296</td>
<td>+ / -</td>
<td>3.08</td>
<td>829231</td>
<td>0.015</td>
<td>94.39</td>
<td>8.27</td>
<td>55.88</td>
</tr>
<tr>
<td>4</td>
<td>500</td>
<td>0.50</td>
<td>0.384</td>
<td>-</td>
<td>1.96</td>
<td>753846</td>
<td>0.014</td>
<td>18.51</td>
<td>2.73</td>
<td>14.22</td>
</tr>
<tr>
<td>5</td>
<td>500</td>
<td>0.50</td>
<td>0.520</td>
<td>+</td>
<td>2.65</td>
<td>1019231</td>
<td>0.014</td>
<td>18.51</td>
<td>5.01</td>
<td>19.25</td>
</tr>
<tr>
<td>6</td>
<td>300</td>
<td>0.25</td>
<td>0.270</td>
<td>+</td>
<td>5.50</td>
<td>1057692</td>
<td>0.016</td>
<td>406.13</td>
<td>29.61</td>
<td>219.31</td>
</tr>
<tr>
<td>7</td>
<td>500</td>
<td>0.40</td>
<td>0.680</td>
<td>-</td>
<td>5.41</td>
<td>1664615</td>
<td>0.014</td>
<td>56.48</td>
<td>26.12</td>
<td>76.81</td>
</tr>
</tbody>
</table>

\[
\Delta Q_{L1} = \frac{-0.21}{91.21} = -0.002 \text{ m}^3/\text{s}
\]
\[
\Delta Q_{LII} = \frac{-0.23}{371.25} = -0.001 \text{ m}^3/\text{s}
\]

The law of energy conservation is fulfilled by allowed discrepancy and correction flow rates are comparatively small. The law of conservation of energy for contour-loop also fulfils:

\[
\sum_{C} H_{L} = -H_{L1} + H_{L2} + H_{L5} + H_{L6} - H_{L7} - H_{L4} = -6.83 + 1.5 + 5.01 + 29.61 - 26.12 - 2.73 = 0.44 \text{ m}.
\]

The result is shown in Fig.4.4.9, the directions of flow and flow rates for each pipe are found, and the system satisfies laws of mass and energy conservation.

Figure 4.4.9. Final solution of the pipe network calculation.
4.5. Minor hydraulic losses

Other hydraulic losses are generally categorized as «minor» or «local» hydraulic losses. Minor hydraulic losses result when there is a significant change in flow pattern and these losses should be evaluated in addition to the friction losses calculated for the pipe length for every pipe system. Minor hydraulic losses can be defined as such type hydraulic losses that are caused by: change of the active cross-section area, change of the active cross-section shape or change of the direction of the flow.

Minor hydraulic losses occur in:

- Pipe entrance or exit (sudden contraction and expansion)
- Fittings (elbows, bends, tees, tapers, connectors, etc.)
- Valves

Minor losses depend on the geometrical peculiarities of the pipe fittings and valves, and it is considered that minor losses can differ depending on particular manufacturers’ products. Therefore the minor loss values listed in this chapter must be taken as averaged values and manufacturers’ data should be used wherever possible.

In some cases minor hydraulic losses can be quite substantial and may not be so minor, for example, relative shorter piping system with various fittings and valves in a pump station. In such part of the piping system the hydraulic losses in fittings and valves can be greater than the losses due to pipe friction.

Since theoretical considerations of the minor losses are quite complex, only some minor losses can be grounded theoretically. Commonly these losses are measured experimentally, correlated with the pipe flow parameters, and expressed by empirical relationships.

Minor losses $h_m$ are commonly expressed in terms of the velocity head as follows:

$$h_m = K \cdot \frac{V^2}{2g}$$  \hspace{1cm} (4.5.1)

where $h_m$ – minor losses, m; $K$ – minor loss coefficient; $V$ – averaged flow velocity, m/s; $g$ – gravitational acceleration, m/s$^2$.

For circular pipes minor losses can be calculated also as: $h_m = 0.0827 \cdot KQ^2 / D^4$, where coefficient $0.0827=16/(\pi^2g)$; $Q$ – flow rate, m$^3$/s; $D$ – diameter, m.

Minor loss coefficient $K$ is dimensionless. Coefficient $K$ is usually showed for turbulent flow conditions and correlated to the geometrical size of the pipe/fitting/valve opening. Experimental studies for turbulent flow at large Reynolds numbers $Re$ showed that $Re$ number have insignificant influence on coefficient $K$ therefore it can be accepted that $Re$ is ignored. However, coefficient $K$ strongly depends on $Re$ for laminar flow.

Another method used is called equivalent length method where minor losses are associated with the particular length of a pipe with same resistance as fitting/valve. Minor losses then are calculated using Darcy-Weisbach equation:

$$h_m = f \cdot \frac{L_e}{D} \cdot \frac{V^2}{2g}$$  \hspace{1cm} (4.5.2)
where \( L_e \) – equivalent length, m; \( f \) – friction factor; \( D \) – diameter, m; \( V^2/2g \) – velocity head, m.

Here minor loss coefficient is defined as \( K = f L_e/D \). Equivalent length \( L_e \) values are experimentally determined for a particular fitting or valve. Experimental studies shows almost constant ratio of \( L_e/D \) for a range of type and diameters of the fitting, which can be found in tables in many literature sources.

There can be many minor losses in the particular pipe system. Minor losses can be counted up for the pipes with equal diameter (equal flow velocity):

\[
\Sigma h_m = \Sigma K \cdot \frac{V^2}{2g}
\]  

(4.5.3)

Minor losses should be counted up separately for a pipe systems with different diameters:

\[
\Sigma h_m = \Sigma K_1 \cdot \frac{V_1^2}{2g} + \Sigma K_2 \cdot \frac{V_2^2}{2g} + \Sigma K_n \cdot \frac{V_n^2}{2g}
\]  

(4.5.4)

4.5.1. Minor hydraulic losses in sudden expansion (Borda-Carnot equation)

Sudden expansion of the pipe system, e.g., pipeline outlet into the reservoir or sudden increase in the pipe diameter, can be in the best way described by theory. In such cases the impact of flow masses occurs where following mass of flow with greater flow speed runs into the flow mass with less flow speed in the expansion. Mixing of the flow masses and generation of the vortices in the secondary flow results in the loss of energy.

Cross-sections I-I and II-II are considered to compare for a sudden expansion (see Fig. 4.5.1). Shear stress of separated flow is insignificant in expansion therefore we use section II-II at the end of separation zone. Flow mass discharge through section I-I in time \( dt \) is defined as \( \rho Q_1 dt = \rho A_1 V_1 dt \), but same amount discharge through section II-II is \( \rho Q_2 dt = \rho A_2 V_2 dt \). These mass momentums differs because of \( V_1 > V_2 \).

![Figure 4.5.1. Sudden expansion of pipe](image)

Increase of momentum in the flow direction:

\[
\rho A_2 V_2 dt V_2 - \rho A_1 V_1 dt V_1 = \rho dt \left( V_2^2 - A_1 V_1^2 \right)
\]  

(4.5.5)
Based on the theorem that change of momentum is equal to the applied force we can use definition of force \( p_1A_2 \) acting on the flow mass in the flow direction and force \( p_2A_2 \) acting in the opposite direction to equate resulting momentum and force:

\[
\rho dt \left( p_2V_2^2 - A_1V_1^2 - \Phi_1 - p_2A_2 dt \right)
\]

(4.5.6)

where \( p_1 \) and \( p_2 \) – hydrostatic pressure assumed equally distributed in section I-I and II-II, respectively, Pa.

By dividing (4.5.6) with \( \rho gA_2 dt \) we get:

\[
\frac{p_1 - p_2}{\rho g} = \frac{V_2^2 - V_1^2}{g} = \frac{A_1}{A_2} = \frac{V_2^2 - V_1V_2}{g}
\]

(4.5.7)

where \( V_2 = V_1(A_1/A_2) \) (expressed from continuity equation \( V_1A_1 = V_2A_2 \)).

Bernoulli equation for real flow in two cross sections I-I and II-II (here and further we use uniform flow assuming that kinetic energy coefficient \( \alpha = 1 \)):

\[
z_1 + \frac{p_1}{\rho g} + \frac{V_1^2}{2g} = z_2 + \frac{p_2}{\rho g} + \frac{V_2^2}{2g} + h_m
\]

(4.5.8)

Assuming \( z_1 = z_2 \) for horizontal pipe layout and using (4.5.7) we can express \( h_m \) from (4.5.8):

\[
h_m = \frac{V_2^2 - V_1V_2}{g} + \frac{V_1^2 - V_2^2}{2g} = \frac{2V_2^2 - 2V_1V_2 + V_1^2 - V_2^2}{2g} = \frac{\Phi_1 - V_2^2}{2g}
\]

(4.5.9)

Equation (4.5.9) is called Borda-Carnot equation. According to Borda-Carnot theorem the pressure head loss in sudden expansion is equal to the velocity head responding for the lost velocity \( (V_1 - V_2) \).

Using expression \( V_2 = V_1(A_1/A_2) \) or \( V_1 = V_2(A_2/A_1) \) and (4.5.9) we get minor losses in the pipe expansion:

\[
h_m = \left( 1 - \frac{A_1}{A_2} \right)^2 \cdot \frac{V_2^2}{2g} = \left( \frac{A_2}{A_1} - 1 \right)^2 \cdot \frac{V_2^2}{2g}
\]

(4.5.10)

where according to (4.5.1) we get \( K_1 = (1 - (A_1/A_2))^2 \) associated to velocity \( V_1 \) or \( K_2 = (A_2/A_1 - 1)^2 \) associated to the velocity \( V_2 \).

For flow discharge into the large reservoir or so called pipe exit (see Fig.4.5.2) it can be assumed that \( V_2 = 0 \) or \( A_2 = \infty \).
By using (4.5.9) and \( V_2 = 0 \) we get:

\[
h_m = \frac{V_i^2}{2g}
\]  

(4.5.11)

This means that all kinetic energy is lost when flow stops in reservoir.

We can express loss coefficient in any type of pipe exit (sharp, bevel, rounded) as follows:

\[
K_I = \left(1 - \frac{A_i}{A_\infty}\right)^2 = 1
\]  

(4.5.12)

### 4.5.2. Minor hydraulic losses in sudden contraction

Sudden contraction can be discussed when there is a pipe entrance from reservoir or sudden decrease in the pipe diameter. Pipe entrances can be defined as: pulled out (Fig. 4.5.3), sharp (Fig. 4.5.4a), bevel (Fig. 4.5.4b), rounded (Fig. 4.5.4c).

For pipe entrance from reservoir (Fig. 4.5.3) there is theoretical explanation when \( L > 0.5D_2 \). There can be distinguished: reducing flow part (active flow cross-section changes from \( A_i \) to \( A_C \)) and expanding flow part (active flow cross-section changes from \( A_C \) to \( A_2 \)). Active flow cross-section reaches minimum equal to \( A_C \) because of the flow separation and secondary flow circulation; this is called \textit{vena contracta} and can be defined by contraction coefficient \( \varepsilon = \frac{A_C}{A_2} \). According to experiments most significant minor losses occurs in the flow expansion part. Therefore flows velocity \( V_C = \frac{Q}{A_C} \) is used in Borda-Carnot equation and we get:

\[
h_m = \frac{\frac{V_i}{V_2} - 1}{2g} = \left(\frac{V_C}{V_2} - 1\right) \frac{V_2^2}{2g} = \left(\frac{A_2}{A_C} - 1\right) \frac{V_2^2}{2g}
\]  

(4.5.13)

where loss coefficient \( K = ((A_2/A_C) - 1)^2 = (1/\varepsilon - 1)^2 \) and contraction coefficient \( \varepsilon \) is found experimentally. Loss coefficient is also expressed as \( K = 1 - (A_2/A_i) \).
Other sudden contraction types (Fig. 4.5.4) gives less vena contracta and thus less minor hydraulic losses. General equation for minor losses is:

\[ h_m = K_C \frac{V_2^2}{2g} \]  

(4.5.14)

where \( K_C \) – loss coefficient of contraction equal to:

\[ K_C = K \left( 1 - \frac{A_2}{A_1} \right) \]  

(4.5.15)

and here \( K \) is defined as coefficient of reducing contraction (obtained experimentally).

For a sharp pipe entrance (Fig. 4.5.4a) \( K=0.5 \) which gives \( K_C=0.5(1-(A_2/A_1)) \).

For a bevel pipe entrance (Fig. 4.5.4b) coefficient \( K \) depends from angle \( \theta \) and relation of \( L/D_2 \) according to Fig. 4.5.5 (Чугаев, 1982).

For a rounded pipe entrance (Fig. 4.5.4c) coefficient \( K \) depends from relation \( r/D_2 \). Coefficient \( K \) reaches minimum \( K=0.03 \) at \( r/D_2=0.2 \) in Figure 4.5.6 (Чугаев, 1982).
4.5.3. **Minor hydraulic losses in curve**

Minor hydraulic losses in elbows or bends can be discussed wherever there is a sharp change of the flow direction (Fig. 4.5.7) or there is a conduit curved at a certain angle (Fig. 4.5.8). Losses are much higher than those at a straight pipe because of the flow separation; secondary flow circulation and *vena contracta* in the curved flow.

Because of the active flow cross-section reduction (Fig. 4.5.7) minor losses in sharp elbow:

\[
h_m = \left( \frac{C}{2g} \right)^2 = \left( \frac{V_c - V}{\varepsilon} \right)^2 = \left( \frac{l}{\varepsilon - l} \right)^2 \frac{V^2}{2g}
\]  \hspace{1cm} (4.5.16)

where loss coefficient \( K = (l/\varepsilon - l)^2 \).
Table 4.5.1. $K$ depending from pipe diameter $D$

<table>
<thead>
<tr>
<th>$D$, mm</th>
<th>20</th>
<th>25</th>
<th>34</th>
<th>39</th>
<th>49</th>
</tr>
</thead>
<tbody>
<tr>
<td>$K_{90^\circ}$</td>
<td>1.7</td>
<td>1.3</td>
<td>1.1</td>
<td>1.0</td>
<td>0.83</td>
</tr>
</tbody>
</table>

Table 4.5.2. $K$ depending from angle $\theta$

<table>
<thead>
<tr>
<th>$\theta$, $^\circ$</th>
<th>20</th>
<th>40</th>
<th>60</th>
<th>80</th>
<th>90</th>
<th>100</th>
<th>120</th>
<th>140</th>
</tr>
</thead>
<tbody>
<tr>
<td>$K_{\theta}$</td>
<td>0.045</td>
<td>0.14</td>
<td>0.36</td>
<td>0.74</td>
<td>0.98</td>
<td>1.26</td>
<td>1.86</td>
<td>2.43</td>
</tr>
</tbody>
</table>

Contraction coefficient $\varepsilon = A_C/A$ depends from elbow angle $\theta$ and pipe diameter $D$, and it is obtained experimentally. Some $K$ values for elbows with $\theta = 90^\circ$ are showed for different pipe diameters in Table 4.5.1 (Чугаев, 1982). For other angles it is used $K_{\theta} = K_{90^\circ}(1 - \cos \theta)$. Averaged values of $K_{\theta}$ for different angles are showed in Table 4.5.2 (Чугаев, 1982).

For curved pipes or bends (Fig. 4.5.8) minor loss is dependant from friction factor $f$, angle $\theta$, ratio of $D/r$. For $Re \geq 2 \cdot 10^5$ minor losses can be found according to (4.5.1) where $K$:

$$K = K' \cdot \frac{\theta}{90^\circ}$$  \hspace{1cm} (4.5.17)

where loss coefficient $K'$ is taken from Table 4.5.3.

Table 4.5.3. Loss coefficient $K'$ for equation 4.5.17 (Чугаев, 1982)

<table>
<thead>
<tr>
<th>$D/2r$</th>
<th>0.1</th>
<th>0.2</th>
<th>0.3</th>
<th>0.4</th>
<th>0.5</th>
<th>0.6</th>
<th>0.7</th>
<th>0.8</th>
<th>0.9</th>
<th>1.0</th>
</tr>
</thead>
<tbody>
<tr>
<td>$K'$</td>
<td>0.13</td>
<td>0.14</td>
<td>0.16</td>
<td>0.21</td>
<td>0.29</td>
<td>0.44</td>
<td>0.66</td>
<td>0.98</td>
<td>1.41</td>
<td>1.98</td>
</tr>
</tbody>
</table>

### 4.5.4. Minor hydraulic losses in tee

For inflow tee (Fig. 4.5.9) we assume $A_1 = A_2$. Loss coefficients $K_{2-3}$ and $K'_{2-3}$ accounting minor losses $h_m(2-3)$ for flow from section II-II to III-III:

$$K_{2-3} = \frac{h_m(2-3)}{V_3^2/2g}; \quad K'_{2-3} = \frac{h_m(2-3)}{V_2^2/2g} = \frac{K_{2-3}}{\left(\frac{A_3}{A_2}\right)^2}$$ \hspace{1cm} (4.5.18)

where $K_{2-3}$ can be found depending from $A_2/A_3$ and $Q_2/Q_3$ (Fig. 4.5.11) (Чугаев, 1982).

Loss coefficients $K_{1-3}$ and $K'_{1-3}$ accounting minor losses $h_m(1-3)$ for flow from section I-I to III-III:
where $K_{1-3}$ can be found depending from $Q_2/Q_3$ (Fig.4.5.12) (Чугаев, 1982).

For outflow tee (Fig.4.5.10) we assume $A_1=A_2$. Loss coefficients $K_{1-2}$ and $K'_{1-2}$ accounting minor losses $h_{m(1-2)}$ for flow from section I-I to II-II:

$$K_{1-2} = \frac{h_{m(1-2)}}{V_1^2/2g} ; \quad K'_{1-2} = \frac{h_{m(1-2)}}{V_2^2/2g} = \frac{K_{1-2}}{\left(1 - \frac{Q_2}{Q_1}\right)^2}$$

(4.5.20)

where $K_{1-2}$ can be found depending from $A_2/A_1$ and $Q_2/Q_1$ (Fig.4.5.13) (Чугаев, 1982).

Loss coefficients $K_{1-3}$ and $K'_{1-3}$ accounting minor losses $h_{m(1-3)}$ for flow from section I-I to III-III:

$$K_{1-3} = \frac{h_{m(1-3)}}{V_1^2/2g} ; \quad K'_{1-3} = \frac{h_{m(1-3)}}{V_3^2/2g} = \frac{K_{1-3}}{\left(1 - \frac{Q_2}{Q_1}\right)^2}$$

(4.5.21)

where $K_{1-3}$ can be found depending from $Q_2/Q_1$ (Fig.4.5.14) (Чугаев, 1982).
4.5.5. **Minor hydraulic losses in gradual contraction**

For gradual contraction or gradual decrease of a pipe diameter there are used such fittings as *tapers*. The energy loss in gradual contraction is less than in sudden contraction. Minor losses in gradual contraction can be found similar like in the case with sudden contraction or bevelled pipe entrance (Fig. 4.5.4b) at a certain contraction angle $\theta$ according to (4.5.14) and (4.5.15):

$$h_m = K \cdot \left(1 - \frac{A_2}{A_1}\right) \frac{V_2^2}{2g} \quad (4.5.22)$$

where $K$ – coefficient of reducing sudden contraction. Coefficient $K$ depends from angle $\theta$ and relation of $L/D_2$ according to Fig. 4.5.5.
4.5.6. Minor hydraulic losses in gradual expansion

Gradual conical expansion fitting or so-called diffuser is used in pipe systems to reduce minor losses in places where pipe diameters are increased (Fig. 4.5.15). The energy loss is less than those in sudden expansion. Minor losses in diffuser depend from angle of expansion and pipe diameter. At angles $8...10^\circ < \theta < 50...60^\circ$ there occurs secondary flows in diffuser, and at angles $\theta > 50...60^\circ$ secondary flows occurs all along the diffuser (Fig. 4.5.15).

Figure 4.5.15. Gradual expansion

A minor loss in diffuser $h_{\text{m,diff}}$ is expressed as a part of minor losses in a sudden expansion $h_m$:

$$h_{\text{m,diff}} = \varphi \cdot h_m = \varphi \cdot \frac{\rho \left( V_1 - V_2 \right)^2}{2g}$$  \hspace{1cm} (4.5.23)

where $\varphi$ – empirical angle coefficient; $h_m$ – minor losses calculated by (4.5.9). Empirical angle coefficient $\varphi$ depends from angle $\theta$ of gradual expansion (Fig. 4.5.16). Smallest losses were found at expansion angle $\theta \approx 6^\circ$ according to experimental data.

Figure 4.5.16. Angle coefficient $\varphi$ (Чугаев, 1982)
4.5.7. **Minor hydraulic losses in valves**

There are many types of valve (gate valve, butterfly valve, globe valve, ball valve, check valve, etc.) used in pipeline systems, e.g., water supply systems (Fig. 4.5.17). Minor losses in valves depend on the valve type, opening ratio or angle of valve closing mechanism, pipe diameter, particular manufacturer’s valve design and average flow velocity at the point not affected by the valve. Minor losses in any type of valve from section I-I to II-II (e.g., Fig. 4.5.17) is expressed as:

\[ h_m = K_{valve} \cdot \frac{V^2}{2g} \]  

(4.5.24)

where \( K_{valve} \) – loss coefficient in a particular valve; \( V \) – flow velocity in a pipe, m/s.

![Diagram of valve types](image)

Figure 4.5.17. Common valve types: (a) gate valve, (b) disk-type gate valve, (c) butterfly valve, (d) ball valve, (e) globe valve, (f) swing-type check valve

<table>
<thead>
<tr>
<th>Valve type:</th>
<th>Nominal diameter DN, mm</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>15</td>
</tr>
<tr>
<td>Gate valve</td>
<td>0.22</td>
</tr>
<tr>
<td>Butterfly valve</td>
<td>-</td>
</tr>
<tr>
<td>Ball valve</td>
<td>0.08</td>
</tr>
<tr>
<td>Globe valve</td>
<td>9.2</td>
</tr>
<tr>
<td>Swing check valve</td>
<td>2.7</td>
</tr>
</tbody>
</table>
In many literature sources and manufacturer’s reports experimental results for \( K_{valve} \) loss coefficient can be found. For gate valve (Fig.4.5.17a,b) \( K_{valve} \) depends from opening ratio \( h/D \) and diameter \( D \), for butterfly valve (Fig.4.5.17c), ball valve (Fig.4.5.17d) and swing type check valve (Fig.4.5.17f) \( K_{valve} \) depends from diameter \( D \) and angle of valve closing mechanism \( \theta \), and for globe valve (Fig.4.5.17e) \( K_{valve} \) depends from opening ratio \( h/D \) and diameter \( D \).

Table 4.5.4 shows minor loss coefficient \( K_{valve} \) for 5 types of fully open valve of a particular manufacturer therefore it must be noted that listed values are somewhat averaged indicator which can vary with uncertainty depending of the geometrical parameters of the valve design (White, 2011). It can be seen that the globe valve gives largest minor losses because of a twisty flow path through valve body. Losses in the partially closed valves are much higher than in the open valves and can be found according to experimental data.

The valve throttling can cause local increase in the pressure losses. Valve throttling is widely used in the water supply systems, e.g., to equalize or alter pressure and vary the flow rate.

4.5.8. **Minor hydraulic losses and Reynolds number**

Minor hydraulic loss coefficient showed in previous chapters is supposed for turbulent flow regime with large Reynolds numbers \( Re \), where impact of the molecular viscosity is negligible.

In the laminar flow or very close to such flow regime, the minor loss coefficient depends from the Reynolds number according to experimental results and empirical relations for some fittings and valves showed in the basic text books.

At the low Reynolds numbers the effect of losses is induced by the viscosity forces and is proportional to the velocity head. In such case, the minor loss coefficient is conversely proportional to \( Re \):

\[
K = \frac{A}{Re}
\]  

(4.5.25)

where \( A \) – a constant depending on the type of the minor loss cause.

At a rather large Reynolds number a separation of the secondary flows occurs which is the base cause of minor losses. In such case \( K = \text{const} \).
In approximation it can be defined that at \( \text{Re} > 3000 \) the loss coefficient \( K \) is not dependant from \( \text{Re} \) for fittings and valves with sharp changes in the active flow cross-section, and at \( \text{Re} > 10000 \) for smooth changes (Калинин, 2007).

The minor loss coefficient in general form is:

\[
K = \frac{A}{\text{Re}} + K_{\text{turb}}
\]  

(4.5.26)

where \( K_{\text{turb}} \) – loss coefficient at large \( \text{Re} \) numbers, when \( K = \text{const} \).

The equivalent length method (Eq.4.5.2) is another method which uses \( \text{Re} \) number as a dependant for loss coefficient estimation, where equivalent length \( L_e \) in the Darcy-Weisbach equation is used. Here a minor resistance is understood as a resistance in the straight pipe with equivalent length.

### 4.5.9. Practical application example

**Example 1.** Determine minor hydraulic losses when 1.3 l/s of water moved from pipe DN 25 mm to pipe DN 50 trough diffuser with angle of expansion \( \theta = 45^\circ \). Pipe specification: material PEH, PN 10, inside diameter DN25 (20.4 mm) and DN50 (40.8 mm). Finally find pressure difference between these two pipes.

**Solution:** find flow velocities, empirical angle coefficient \( \varphi \) and then minor hydraulic losses in diffuser \( h_{m,diff} \).

\( D_1 = 20.4 \text{mm} \ (A_1 = 0.000327 \text{m}^2) \) and \( D_2 = 40.8 \text{mm} \ (A_2 = 0.001307 \text{m}^2) \).

\[
V_1 = \frac{Q}{A_1} = 1.3 \cdot \frac{0.001}{0.000327} = 3.97 \text{ m/s}
\]

\[
V_2 = \frac{Q}{A_2} = 1.3 \cdot \frac{0.001}{0.001307} = 0.99 \text{ m/s}
\]

Use Fig.16 and \( \theta = 45^\circ \) to find \( \varphi = 1.09 \).

Therefore \( h_{m,diff} = \varphi \cdot h_m = \varphi \cdot \left( \frac{V_1^2 - V_2^2}{2g} \right) = 1.09 \cdot \left( \frac{3.97 - 0.99}{2 \cdot 9.81} \right) = 0.49 \text{ m} \)

Use general energy equation considering only minor losses because of short pipe length:

\[
Z_1 + \frac{P_1}{\rho g} + \frac{V_1^2}{2g} - h_{m,diff} = Z_2 + \frac{P_2}{\rho g} + \frac{V_2^2}{2g}
\]

Rearrange equation and solve for \( P_1-P_2 \):

\[
P_1 - P_2 = \rho g \cdot \left( (Z_2 - Z_1) + \frac{(V_2^2 - V_1^2)}{2g} + h_{m,diff} \right)
\]

\[
P_1 - P_2 = 1000 \cdot 9.81 \cdot \left( 0 + (0.99^2 - 3.97^2) / 2 \cdot 9.81 + 0.49 \right) = -2583.5 \text{ Pa} \approx -2.6 \text{ kPa}
\]

This means that \( P_2 > P_1 \) as pressure \( P_2 = P_1 + 2.6 \text{ kPa} \) in pipe DN50.
5. Flow in open channels

Boriss Gjunsburgs
Riga Technical University, Water Engineering and Technology Department
Introduction

Flow in open channels, shape and slope of the free surface, depth are depending on size, shape of the cross section and slope of the channel. The flow in open channel can be:

- uniform and non-uniform;
- steady or unsteady;
- steady uniform or steady non-uniform;
- gradually varied;
- rapidly varied;
- laminar or turbulent;
- tranquil or rapid;
- subcritical or supercritical.

The channels are classified:

- prismatic;
- non-prismatic;
- with slopes \((i > 0, i = 0, i < 0)\).

At uniform turbulent flow the roughness depth of the flow, shape cross section area, wetted perimeter and hydraulic radius are constant. The channel is prismatic with direct slope of channel bed \(i > 0\).

At non-uniform flow results in changes of the cross section area.

The steady flow is when flow velocities are not changing in time: \(V = V(y, z)\).

The unsteady flow is when flow velocities are changing in time: \(V = V(y, z, t)\).

The steady uniform flow when average velocity and depth of the channel are not changing from one section to another.

The steady non-uniform flow is when discharge is not changing in time, but depth and discharge change from one section to another.

The gradual changes in depth or cross section area called gradually varied flow.

The prismatic channel is when main geometrical parameters along the channel are not changing.

The cross section area of the channel depends on the depth: \(\omega = f(h)\).
For non-prismatic channel the cross section area of the channel depends on \( \omega = f(h, b) \), where \( h \) is a depth of the flow and \( b \) is a lateral size of the channel.

The critical Reynolds for flow in pipes is \( Re_{cr} = 2320 \) and hydraulic radius is \( R = d/4 \). For open flow the critical Reynolds’s number is with \( D = 4R \) and then \( Re_{cr} = 500 \), as

\[
Re_{cr} = \frac{V \cdot R}{\nu},
\]

and with \( Re > 500 \) the flow in open channel is turbulent.

The flow is tranquil when Froude number \( Fr = \frac{V}{\sqrt{gh}} \) is less than 1, and the flow is rapid if Froude number is more than 1.

### 5.1. Uniform steady flow in open channels. Basic equations

At uniform steady flow in open channel the piezometric line \( P - P \) coincides with surface profile of water, and flow motion take place with normal depth \( h_0 \) and hydraulic slope \( s \) is equal to piezometric \( I_p \) and channel bed slope \( i_0 \).

![Uniform steady flow in open channel](image)

The hydraulic linear losses can be found from Darcy-Weisbach formula for turbulent flow with hydraulically rough surface:

\[
H_L = \lambda \frac{l}{R} \frac{V^2}{2g}
\]  
\[(5.1.1)\]
where $H_L$ – hydraulic linear losses; $\lambda$ - friction coefficient of the channel; $R$ – hydraulic radius ($R = \frac{D}{4}$); $V$ – average velocity of the flow; $g$ – gravitational acceleration.

The velocity from Eq.(5.1.1) applied for open flow

$$V = \sqrt{\frac{8g}{\lambda} \sqrt{R \frac{H_L}{l}}} \quad (5.1.2)$$

or

$$V = C \sqrt{R \cdot i_0} \quad (5.1.3)$$

In Chezy formula (Eq.5.1.3) $C$ is Chezy coefficient $C = \sqrt{\frac{8g}{\lambda}}$ and is dependent on friction coefficient, flow regime and roughness. Chezy coefficient is supposed to be constant for any size of the cross section and roughness of the channel. Chezy coefficient can be obtained experimentally and nowadays can be found by various formulas.

Manning formula:

$$C = \frac{R^{1/6}}{n} \quad (5.1.4)$$

where $n$ is a roughness factor (Table 5.1).

Table 5.1. Roughness factor for various materials

<table>
<thead>
<tr>
<th>Type of the channel</th>
<th>$n$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Sand, without vegetation</td>
<td>0.02 – 0.03</td>
</tr>
<tr>
<td>Sand, with bed sediment movement, floodplain</td>
<td>0.023 – 0.033</td>
</tr>
<tr>
<td>Concrete</td>
<td>0.012 – 0.014</td>
</tr>
<tr>
<td>Metal</td>
<td>0.018 – 0.022</td>
</tr>
<tr>
<td>Cast iron</td>
<td>0.015</td>
</tr>
<tr>
<td>Brick</td>
<td>0.016</td>
</tr>
<tr>
<td>Gravel</td>
<td>0.029</td>
</tr>
</tbody>
</table>

Pavlovsky formula:

$$C = \frac{R^{1/6}}{n} \quad (5.1.5)$$
where \( y = f(R, n) \) at \( R > 1 \text{m} \), \( y = 1.5 \) and at \( R < 1 \text{m} \), \( y = 1.3 \).

Chezy formula is used to find discharge at the uniform flow

\[
Q = \omega V = \omega C \sqrt{R \cdot i_0}
\]  
(5.1.6)

where \( \omega \) – cross section area.

Eq.(5.1.6) can be presented as:

\[
Q = K \sqrt{i_0}
\]  
(5.1.7)

where \( K \) – discharge module \( K = \omega C \sqrt{R} \) and is equal to the discharge at slope \( i_0 = 1 \).

In Eq.(5.1.6) \( C = \frac{R^y}{n} \) and \( R = \frac{\omega}{\chi} \), then

\[
Q = \omega \frac{R^y}{n} R^{1/2} i_0^{1/2}
\]  
(5.1.8)

and the basic formula for channel calculation at uniform steady flow is

\[
Q = \omega^{3/2+y} \frac{1}{\chi^{1+y}} \frac{i_0^{1/2}}{n}
\]  
(5.1.9)

Next solution can be made using Eq.(5.1.9) to calculate the uniform steady flow and parameters of the cross section:

1. Calculate possible discharge through fixed cross section area with known channel parameters.

For determination of the discharge for fixed cross section is given:

- depth of the flow \( h_0 \), with of the cross section \( b \), slope of the side walls \( m \);
- slope of the channel \( i_0 \), roughness factor \( n \);
- cross section area and wetted perimeter.

Solution: Use Eq.(5.1.9) for direct calculation of discharge \( Q \).

2. Appoint the slope of the channel.

For calculation the slope of the channel is given:

- Discharge \( Q \);
• Dimensions of the cross section–\(h_0, b, m = \text{ctg} \alpha\).

Solution: The discharge module is equal to \(K = \omega C \sqrt{R}\), and discharge is found by Eq.(5.1.7). The slope of the channel is:

\[
i_0 = \frac{Q^2}{K}
\]  \hspace{1cm} (5.1.10)

3. Calculate hydraulic best cross section area and \(b/h\) ratio.

Not known the cross section parameters \(b\) or \(h_0\), and is necessary to find ratio \(b / h_0\) and best hydraulic channel cross section. The cross sections of the channel \(\omega\) with minimal wetted perimeter \(\chi_{\text{min}}\) and with maximal value of the hydraulic radius \(R_{\text{max}}\) have maximal capacity for the discharge (see Eq.5.1.9) and are called as best hydraulic channel cross section. The discharge is maximal when wetted perimeter is \(\chi_{\text{min}}(Q_{\text{max}} \rightarrow \chi_{\text{min}})\).

It is known that for given cross section the best wetted perimeter has circle or half of the circle (Fig.5.1.2)

![Figure 5.1.2. Best hydraulic cross section of the channel](image)

Hydraulic radius for this cross section:

\[
R = \frac{\omega}{\chi} = \frac{\frac{\pi D^2}{8}}{\frac{\pi D}{4}} = \frac{D}{4} = \frac{h_0}{2}
\]  \hspace{1cm} (5.1.11)

As construction of the half-circle cross section in practice is complicated, the ratio \(b / h_0\) and \(\chi_{\text{min}}\) will be determined for trapezoidal cross section (Fig.5.1.3), where \(h_0\) – normal depth, \(b\) – with of the channel bed, \(m\) – slope of the side walls \((m = \text{tg} \alpha)\).
Figure 5.1.3. Trapezoidal cross section of the channel

Solution:

1) The wetted perimeter $\chi$ is: $\chi = b + 2h_0 \sqrt{1 + m^2}$ \hfill (5.1.12)

2) Cross section area: $\omega = b \cdot h_0 + mh_0^2$ \hfill (5.1.13)

3) Width of the channel: $b = \frac{\omega - mh_0^2}{h_0}$ \hfill (5.1.14)

4) Substituting $b$ in formula for wetted perimeter: $\chi = \frac{\omega - h_0^2}{h_0} + 2h_0 \sqrt{1 + m^2}$ \hfill (5.1.15)

The wetted perimeter is a function $h_0$ and minimal value will be, when derivative is equal to zero. Differentiable:

$$\frac{d\chi}{dh_0} = 2\sqrt{1 + m^2} - 2m - \frac{\omega - h_0^3}{h_0^2} = 0$$ \hfill (5.1.16)

$$\frac{\omega - mh_0^2}{h_0^2} = 2\sqrt{1 + m^2} - 2m$$ \hfill (5.1.17)

$$\left(\frac{b}{h_0}\right) = 2\sqrt{1 + m^2} - 2m$$ \hfill (5.1.18)

where $m$– slope of the side wall (Table 5.2).

Table 2. Dependence of the width to the depth of the channel on the wall slope for the best hydraulic cross section area.

<table>
<thead>
<tr>
<th>$m$</th>
<th>1</th>
<th>1.5</th>
<th>2</th>
</tr>
</thead>
<tbody>
<tr>
<td>$b / h_0$</td>
<td>0.828</td>
<td>0.606</td>
<td>0.472</td>
</tr>
</tbody>
</table>
5.2. Normal depth of the flow in channel

Depth in channel at uniform steady flow is called normal depth \( h_0 \). Discharge at the normal depth is

\[
Q = \omega_0 C_0 \sqrt{R_0 i_0} \text{ or } Q = K_0 \sqrt{i_0}
\]  \hspace{1cm} (5.2.1)

where parameters \( \omega_0, C_0, R_0 \) and \( K_0 \) at normal depth in channel.

The normal depth can be found graphically.

![Diagram of normal depth determination](image)

Figure 5.2.1. Normal depth in channel determination

Solution:

For given cross section area the discharge modules \( K_1, K_2, ..., K_n \) for different depth of the flow must be calculated

\[
K = \omega C \sqrt{R}
\]  \hspace{1cm} (5.2.2)

Dependence of the depths to discharge modules can be presented graphically.

Discharge module at normal depth:

\[
K_0 = \frac{Q}{\sqrt{i_0}}
\]  \hspace{1cm} (5.2.3)

Normal depth can be found graphically (Fig.5.2.1).

5.3. Specific energy of the cross section. Critical depth

Total specific energy for cross section in open flow compare to the datum plane is:
\[ H = z + \frac{P}{\rho g} + \frac{\alpha V^2}{2g} \]  

(5.3.1)

\[ E = h + \frac{\alpha V^2}{2g} = h + \frac{\alpha Q^2}{2g \omega^2} \]  

(5.3.2)

where velocity \( V = \frac{Q}{gh} = \frac{q}{h} \); and \( q \)– specific discharge. Then:

\[ E = h + \frac{\alpha q^2}{2gh^2} \]  

(5.3.3)

The specific energy of the cross section is a function of the flow depth:

\[
\begin{align*}
    h &\to 0 & E &\to 0 \\
    h &\to \infty & E &\to \infty
\end{align*}
\]

Figure 5.3.3. Specific energy changes versus depth of flow
Depth corresponding to minimum of the specific energy of the cross section is called critical depth.

For determination of the \( E_{\text{min}} \) the derivative should be equate to zero.

\[
\frac{dE}{dh} = 1 - \frac{\alpha Q^2}{g \omega^3} \frac{d\omega}{dh} = 0 \tag{5.3.4}
\]

Figure 5.3.4. Critical depth of flow determination

The Eq.(5.3.4) can be as:

\[
\frac{dE}{dh} = 1 - \frac{\alpha Q^2}{g} \frac{B_{\text{cr}}}{\omega_{\text{cr}}^3} = 0 \tag{5.3.5}
\]

\[
\frac{\alpha Q^2 B_{\text{cr}}}{g \omega_{\text{cr}}^3} = 1 \tag{5.3.6}
\]

or

\[
\frac{\alpha Q^2}{g} = \frac{\omega_{\text{cr}}^3}{B_{\text{cr}}} \tag{5.3.7}
\]

Critical depth can be calculated using Eq.(5.3.7).

For rectangular cross-section of the channel the formula for critical depth is:

\[
h_{\text{cr}} = \sqrt[3]{\frac{\alpha Q^2}{gb^2}} \tag{5.3.8}
\]

For triangular cross-section of the channel the formula for critical depth is:
For trapezoidal channels the critical depth should be found graphically.

Solution:

1) For different depth of flow the ratio \( \omega^3 / B \) is calculated and presented graphically.

\[
h_{cr} = \sqrt[5]{\frac{\alpha Q^2}{gm}}
\] (5.3.9)

\( \h_{cr} \)

2) \( \alpha Q^2 / g \) is calculated

3) Critical depth can be found graphically

In comparison of the depth channel with critical depth, the flow can be classified:

- tranquil, when real depth of uniform or non-uniform flow is greater the critical depth \( h > h_{cr} \);
- rapid, when real depth of flow is less than critical depth \( h < h_{cr} \);
- critical, when real depth of flow is equal to critical one \( h = h_{cr} \).

Another classification of the flow is subcritical and supercritical:

- subcritical is the flow with velocities less than critical;
- supercritical is the flow with velocities greater than critical.

### 5.4. Critical slope

Critical slope at steady uniform flow is when normal depth \( h_0 \) is equal to the critical depth \( h_0 = h_{CR} \).
For given value of the shape and the size of the cross section, roughness, discharge and slope the normal depth is set, but with different slope of the channel the normal depth will be different. The normal depth is depending on the slope at the open channel. Critical velocity is not depending on the slope of the channel.

The formula for the critical depth can be presented as:

\[
\frac{\omega_{CR}^3}{B_{CR}} = \frac{\alpha Q \cdot C_{CR} \sqrt{R_{CR} \cdot i_{CR}}}{g}
\]

(5.4.1)

where

\[ Q = \omega_{CR} \cdot C_{CR} \sqrt{R_{CR} \cdot i_{CR}} \]

(5.4.2)

or

\[
\frac{\omega_{CR}}{B_{CR}} = \frac{\alpha \cdot C_{CR}^2 \cdot R_{CR} \cdot i_{CR}}{g}
\]

(5.4.3)

The critical slope from Eq.(5.4.3) is:

\[ i_{CR} = \frac{g}{\alpha C_{CR}^2} \cdot \frac{\omega_{CR}}{B_{CR} \cdot R_{CR}} \]

(5.4.4)

where \( \frac{R_{CR}}{\omega_{CR}} \) is equal \( X_{CR} \).

Then critical slope is:

\[ i_{CR} = \frac{g}{\alpha C_{CR}^2} \cdot \frac{X_{CR}}{B_{CR}} \]

(5.4.5)

5.5. Velocities restriction at open channel design

The bed and walls of the earth open channel will be stable when velocities are equal:

1) at clear water conditions, when there is not sediment movement:

\[ V < V_{MAX} \]  

(5.5.1)
where $V$ – velocity of the flow; $V_{MAX}$ - permitted velocity, is dependent on the soil of the channel is constructed.

2) at bed and suspended sediment movement:

$$V_{\text{min}} < V < V_{MAX}$$  \hspace{1cm} (5.5.2)

where $V_{\text{min}}$ – velocity when sediments deposition from water will not take place.

For some materials the maximal permitted velocity presented in Table 5.5.1.

<table>
<thead>
<tr>
<th>Materials</th>
<th>$V_{\text{perm}} \text{ (m/sec)}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>dust</td>
<td>0.15 – 0.20</td>
</tr>
<tr>
<td>sand</td>
<td>0.20 – 0.60</td>
</tr>
<tr>
<td>gravel</td>
<td>0.60 – 1.20</td>
</tr>
<tr>
<td>clay</td>
<td>1.20 – 1.80</td>
</tr>
<tr>
<td>rocky solids</td>
<td>2.50 – 4.50</td>
</tr>
<tr>
<td>paving</td>
<td>3.00 – 3.50</td>
</tr>
</tbody>
</table>

**5.6. Steady non-uniform gradually varied flow in open channels. Basic differential equation**

**5.6.1. General form of basic differential equation**

Steady non-uniform flow is when depth at the channel and average velocity is changing from one cross section to another (Fig. 5.6.1).

Bernoulli equation for two cross sections is used:

$$z_1 + \frac{p_a}{\gamma} + \frac{\alpha V_1^2}{2g} = z_2 + \frac{p_a}{\gamma} + \frac{\alpha V_2^2}{2g} + h_i$$  \hspace{1cm} (5.6.1)

as $p=p_a$

$$z_1 - z_2 = \frac{\alpha V_2^2 - \alpha V_1^2}{2g} + h_i$$  \hspace{1cm} (5.6.2)
Figure 5.6.1. Steady non-uniform gradually varied flow

Changing distance $S$ between cross sections to infinitely small one $ds$, Eq.(5.6.2) can be presented as:

$$dz = d \frac{\alpha V^2}{2g} + dhl$$

(5.6.3)

The flow movement is gradually varied and head losses then can be calculated as for steady uniform flow

$$\frac{dz}{dS} = \frac{d}{dS} \frac{\alpha V^2}{2g} + \frac{dh_l}{dS}$$

(5.6.4)

Surface of the water in channel is reducing along it, and then $\frac{dz}{dS}$ is negative:

$$-\frac{dz}{dS} = \frac{d}{dS} \frac{\alpha V^2}{2g} + \frac{V^2}{C^2R}$$

(5.6.5)

or according to Chezy formula:

$$I = I_v + I_f$$

(5.6.6)

where $\frac{dz}{dS} = I$ slope of the free surface, $I_v$ – slope of the velocity head; $I_f = \frac{V^2}{C^2R}$ is slope of the friction head.
Eq.(5.6.5) is basic differential for steady non-uniform flow in open channels.

5.6.2. **Second form of the basic differential equation**

The basic differential Eq.(5.6.5) will be converted to the second form.

The value of the first term in Eq.(5.6.5) \(- \frac{dz}{dS}\) can be presented as (Fig. 5.6.2)

\[
\frac{-dz + dh}{dS} = i_0 \tag{5.6.7}
\]

\[
- \frac{dz}{dS} = i_0 - \frac{dh}{dS} \tag{5.6.8}
\]

![Figure 5.6.2. Water surface profile steady non-uniform gradually varied flow](image)

For non-prismatic channel \(w=f(h,s)\):

\[
\frac{d\omega}{dS} = \frac{\partial \omega}{\partial S} + \frac{\partial \omega}{\partial h} \frac{dh}{dS} \tag{5.6.10}
\]

For prismatic channel \(w=f(h)\), \(\frac{\partial \omega}{\partial S} = 0\) and \(\frac{\partial \omega}{\partial h} = B\).

Then equation (5.6.10) is:
\[ \frac{d\omega}{dS} = B \frac{dh}{dS} \quad (5.6.11) \]

and

\[ \frac{d}{dS} \frac{\alpha V^2}{2g} = - \frac{\alpha Q^2}{g} \frac{B}{\omega^3} \quad (5.6.12) \]

The third term of Eq. (5.6.5):

\[ \frac{V^2}{C^2 R} = \frac{Q^2}{\omega^2 C^2 R} \quad (5.6.13) \]

The discharge \( Q \) is not changing along the channel, and at the beginning of the motion there is a cross section where depth of the flow is equal to normal depth \( h = h_0 \)

\[ \frac{V^2}{C^2 R} = \frac{Q^2}{\omega^2 C^2 R} = \frac{w_0 C_0 R_0}{\omega^2 C^2 R} \cdot i_0 = \frac{K_0^2}{K^2} \cdot i_0 \quad (5.6.14) \]

where \( Q = \omega \cdot V = \omega \cdot c \sqrt{Ri_0} = K \sqrt{i_0} \).

\( V \) is determined according to Chezy formula, \( K \) is discharge module.

Equation (5.6.5) can be written as:

\[ i_0 - \frac{dh}{dS} = - \frac{\alpha Q^2}{g} \frac{B}{\omega^3} \frac{dh}{dS} + i_0 \left( \frac{K_0}{K} \right)^2 \quad (5.6.15) \]

or

\[ i_0 \left( 1 - \frac{K_0^2}{K} \right) = \frac{dh}{dS} \left( 1 - \frac{\alpha Q^2}{g} \frac{B}{\omega^3} \right) \quad (5.6.16) \]

Differential equation on steady non-uniform gradually varied flow in prismatic channels can be presented in the second form:

\[ \frac{dh}{dS} = i_0 \frac{1 - \frac{K_0^2}{K^2}}{1 - \frac{\alpha Q^2}{g} \frac{B}{\omega^3}} \quad (5.6.17) \]
5.7. Practical application examples

**Example 1.** Determine flow rate \( Q \) and flow velocity \( V \) in trapezoidal channel with roughness factor \( n=0.025 \), channel slope \( i=0.0002 \), side wall slope \( m=1.25 \), width \( b=10m \), and height \( h_0=3.5m \).

**Solution:** Calculate cross section area (Eq.5.1.13):
\[
\omega = b \cdot h_0 + mh_0^2 = 10 \times 3.5 + 1.25 \times 3.5^2 = 50.3m^2
\]
Calculate wetted perimeter (Eq.5.1.12):
\[
\chi = b + 2h\sqrt{1 + m^2} = 10 + 2 \times 3.5\sqrt{1 + 1.25^2} = 21.2m
\]
Calculate hydraulic radius (Eq.5.1.11):
\[
R = \frac{\omega}{\chi} = \frac{50.3}{21.2} = 2.37m
\]
Calculate Chezy coefficient: \( C = \frac{R^{1/6}}{n} = \frac{2.37^{1/6}}{0.025} = 46.2m^{1/2} / s \)
By Eq.5.1.6 calculate flow rate: \( Q = \omega C\sqrt{R \cdot i} = 50.3 \times 46.2\sqrt{2.37 \times 0.0002} = 50.6m^3 / s \)
6. Flow in wastewater treatment units

Romans Neilands
Riga Technical University, Water Engineering and Technology Department
Introduction

The hydraulic solution represents a reasonable compromise between engineering calculation and accuracy of information. The mechanical quantities such as pressure and velocity are functions of positions and time and are obtained as solutions of the local and temporal fluid motion (Hager, 2010).

In the design of many sewage treatment plants, the hydraulic has frequently been given not sufficient attention. This often leads to severe operational difficulties such as component units overflowing under peak conditions and component units not coping if some are out of service for maintenance purposes.

The particular aims of this chapter were to pay more attention of the importance of flow hydraulic behaviour in constructions of wastewater treatment units regarding the flow and particle performances and to improve the prediction of process lifetime in full scale applications.

This chapter presents the basics of flow hydraulics as applied in the design wastewater treatment plants and sections covered the basics hydraulic head losses, and flow distribution, design of hydraulic elements, and wastewater treatment plant process hydraulics.

6.1. Specific aspects of wastewater flows

Wastewater is typically divided into one of the followings groups:

- Domestic or sanitary. Wastewater only produced in households.
- Municipal wastewater. There domestic wastewater is mixed with effluents from commercial and industrial works.
- Industrial wastewater. Wastewater in which industrial wastes dominates.
- Storm water. Water resulting from precipitation runoff.

The information about mean water consumption is approximately equal to wastewater production per inhabitant per day. The average figures are different from country to country, for example in Latvia for average size towns are in range 125 l per capita.
6.1.1. **Flow regimes**

The municipal daily flow rate often changes by a factor 2 - 4, depending on size of town and industrial flow components, also from the sewer system, there quality, and infiltration/exfiltration rates. The minimum values are in nights and the maximum in mornings and evenings which is typical for municipal wastewater treatment plants. The daily and weekly changes in the flow rate for municipal sewers are given in Figures 6.1.1, 6.1.2.

![Figure 6.1.1. Daily flow variation in municipal wastewater systems (Tchobanoglous et al., 2003)](image1)

![Figure 6.1.2. Weekly flow variation in municipal wastewater systems (Tchobanoglous et al., 2003)](image2)

6.1.2. **Flow regimes in wastewater treatment units**

The wastewater treatment plants utilise unit operations and unit processes to achieve the designed degree of treatment efficiency. In unit operations, the treatment or removal of
contaminants is brought about physical forces. This part is concerned with the hydraulic design of wastewater treatment units. The aims of this part are the treatment unit operation hydraulic principle and processes, which finally lead of the complete hydraulic profile of whole plant. For each treatment unit sections presents unit operation hydraulic aspects and basic of design. Figures 6.1.3 demonstrate the flow regime in wastewater treatment units and distribution system.

The important effect of hydraulics on the performance of wastewater basins has prompted many researchers to study how different design variables influence flow patterns. This usually accomplished by computing parameters such as the mean residence time of the waste stream, and the basin dead volume which quantify the extent of flow non-ideality. These models range from ideal cases such as completely mixed and plug flow reactors to non-ideal.

Therefore, the hydraulic regimes for reactors used in wastewater treatment plants are typically classified as plug-flow, complete-mix, or orbitary flow. Plug flow reactors convey liquid through the tank as plug without longitudinal mixing. Every particle is assumed to remain in the tank for an amount of time (t) equal to V/Q, where V and Q are the tank volume and flowrate, respectively.

6.1.3. **Practical application example - flow rate estimation**

*Estimate the average and maximum hourly flow for a municipality of 1000 persons.*

**Solution:**

1. Estimate wastewater daily flow rate.
Assume average water consumption 125 L/(p d) and that 90% of water consumption goes to the sewer.

Average wastewater flow=125 (l/p d)×0.90×1000 persons×0.001 m³/l = 112.50 m³/d.

2. Compute average hourly flow rate (Q\text{aver}):

Average hourly flow rate (Q\text{aver})=112.50 (m³/d)/24 h = 4.68 m³/h

3. Estimate the maximum hourly flow rate- Q\text{max}

Assume the maximum hourly flow rate is four times the average hourly flow rate, thus:

Maximum hourly flow rate (Q\text{max}) =4.68 m³/h×4=18.72 m³/h

6.2. Bar racks / Screens

The objective of screening is removing large solids from wastewater that may be present so as to protect downstream units from damage, and prevent interference with treatment. A screen is one of the first treatment units in wastewater treatment plant.

6.2.1. Types of screens

Screens are classified as coarse (>6 cm), medium (0.5-6 cm) and fine (0.3-0.6 mm) and microscreens (<0.5μm) and are given in the Figure 6.2.1.

![Figure 6.2.1. Screen classification. Source (Tchobanoglous et al., 2003)](image-url)
6.2.2. **Hydraulic analysis**

Bar racks and screens are commonly found in the headwork of treatment plants and are used to remove large objects from the influent flow. Flow passes through a set of bars, where the flow is constricted and may result in head loss. This head loss is dependent on the size (width of the rack or screen), the spacing of the bars, the width of the bars, and the depth of flow through the rack or screen and angle from vertical. The analysis associated with the determination of the head loss. The losses in screens and bar racks applying the Bernoulli equation before and after the screen as following:

\[
h_1 + \frac{V_1^2}{2g} = h_2 + \frac{V_{sc}^2}{2g} + \Delta h
\]

(6.2.1)

where \(h_1\) – the upstream depth of flow; \(h_2\) – the downstream depths of flow; \(g\) – the acceleration due to gravity; \(V_1\) – the upstream velocity; \(V_2\) – the velocity through the screen.

The difference in depths downstream and upstream of the screening device that results from a change in velocity as flow passes through the screen. For the clean or partially blocked screen, losses are calculated by equation:

\[
\Delta h = h_1 - h_2 = \frac{1}{2gC_d^2} (V_2^2 - V_1^2)
\]

(6.2.2)

where \(C_d\) – discharge coefficient with value of 0.84.

The head loss also can be calculated as an orifice as following equation:

\[
\Delta h = \frac{V_2^2}{2gC_d^2} = \frac{1}{2g} \left( \frac{Q}{C_d \cdot A} \right)
\]

(6.2.3)

where \(A\) – the effective open area of the submerged screen.

To find the velocity through the bars, you must know the free open area of the rack or screen. The free area of the rack or screen by requesting the overall width of the unit, the width of the bars, and the spacing of the bars and then the velocity through the bars can be determined by equation:

\[
V_2 = \frac{Q}{A}
\]

(6.2.4)
The velocity in the approach channel must also be determined. The depth upstream in the channel will be equal to the depth downstream plus the head loss of the unit. But since the head loss is dependent on the upstream depth, the head loss and upstream depth must be determined simultaneously so that both equations correctly work out. The velocity in the upstream channel can then be determined by:

\[ V_1 = \frac{Q}{h_1 \cdot B} \]  

(6.2.5)

where \( B \) – channel width, m.

Once the two velocities are determined, the head loss can then be calculated. It should be noted that the bar spacing (s) must always be greater than the actual width of the bars (b).

When the bars are clean, Kirshmers equation may be used:

\[ \Delta h = \beta \left( \frac{W}{b} \right)^{1.33} \cdot h_1 \cdot \sin \theta, \]  

(6.2.6)

where \( \beta \) – a bar shape factor; \( W \) – total transverse width of bars; \( b \) – total transverse clear spacing between bars; \( h_1 \) – the upstream velocity head; \( \theta \) – the angle of the bars to the horizontal.

The clean water headloss through fine screens may be obtained from manufactures or may be computed by means of orifice equation:

\[ \Delta h = \frac{1}{2g} \left( \frac{V_2}{C} \right)^2 = \frac{1}{2} \left( \frac{Q}{CA} \right)^2 \]  

(6.2.7)

Where \( \Delta h \) – headloss, m; \( V_2 \) – approach velocity, m/s; \( g \) – gravitational acceleration, m/s\(^2\); \( Q \) – discharge through the screen, m\(^3\)/s; \( A \) – area of effective opening of submerged screen, m\(^2\).

### 6.2.3. Design channel depth

The basic design criteria of a bar are approach velocities. The optimum velocity is 0.6 m/s through the screen opening, and maximum 0.75 till 1.0 m/s to prevent entrapped materials being forced through the bars. Design channel depth is the required channel depth to meet a target velocity for flow passing through a screening device. The design
requires minimum flow velocity in upstream channel up to 0.4 m/s, to prevent the
deposition of solids.

These velocities are maintained by downstream weir and the depth is calculated by equation:

\[ h = \left( \frac{Q}{V_1} \right) / B \]  

(6.2.8)

6.2.4. Screenings volume

The quantity of screenings varies with the bar openings, type of screen, wastewater flow and it characteristics. There roughly, can be estimated about 3.0 till 35 L of screenings is produced from 1000 m3 of wastewater treated. The screening volume dependence from the opening between bars is given in the Figure 6.2.2.

![Figure 6.2.2. Screening volume dependency from bar openings](image)

6.2.5. Practical application example - screen design velocity

**Example 1.** The approach velocity of flow in coarse screen channel is 0.6 m/s. The measured headloss is 38 mm. Compute the velocity through a bar rack.

**Solution:** Using the equation (6.2.2) \( \Delta h = \frac{V_{sc}^2 - V_1^2}{0.7 \cdot (2g)} \) estimate the velocity through the bar rack \( V \):
Then $V^2 = 0.882$
Flow velocity through a bar rack is $V = 0.94 \text{ m/s}$

### 6.3. Grit removal

Objective is to remove in organic grit, sand particles 0.2 till 0.35 mm size through differential settling with aim to prevent from damage pumps, blockage of channels, reduce deposition in pipelines and manholes and cementing of sludge in settling tanks. There exist two main types of grit chambers, namely horizontal flow constant or aerated spiral flow tanks and vortex units given in Figure 6.3.1.

![Horizontal flow and vortex type grit tanks](image)

Figure 6.3.1. Horizontal flow and vortex type grit tanks (Tchobanoglous et al., 2003)

The use of CFD programs which is applicable to define flow hydraulic patterns in grit tank and inlet and outlet structures, see Figures 6.3.2.
6.3.1. **Hydraulic analysis**

The design goal is to provide sufficient detention time for grit to settle and pass forward the lighter materials.

6.3.2. **Grit settling velocity**

Grit settling velocity is the time it will take a specified type of grit to settle a determined depth. The settling velocity of discrete particle, as there applied to Class 1 settling theory, and can be determinate using appropriate equation depending upon Reynolds number, the Stokes law which solving fall settling or terminal velocity:

\[
V_s = \frac{g \cdot (\rho_s - \rho_m) \cdot d^2}{18 \cdot \mu}
\]  

(6.3.1)

where \(V_s\) – particle sedimentation velocity, m/s; \(g\) – gravitational acceleration, m/s\(^2\); \(\rho_s\) – density of particle, kg/m\(^3\); \(\rho_m\) – density of medium, kg/m\(^3\); \(\mu\) – viscosity of medium, kg/s.m.

The velocity needed to scour lighter materials such as organics given by Camp-Shields equation:

\[
V_c = \sqrt{\frac{8k \cdot g \cdot d}{f} \cdot \left( \frac{\rho_s - \rho_m}{\rho_m} \right)}
\]  

(6.3.2)

where \(V_c\) – scour velocity, m/s; \(g\) – gravitational acceleration, m/s\(^2\); \(d\) – particle diameter, m; \(f\) – Darcy-Weissbach friction factor; \(k\) – empirical constant related to density of particles, are in range 0.04 till 0.06; \(\rho_s\) – particle density, kg/m\(^3\); \(\rho_m\) – density of medium, kg/m\(^3\).

Typically \(V_c\) are 15 to 39 cm/s for organic particles. To maintain the \(V_c\) through fluctuation of incoming flow rate is used outflow weir to maintain velocity as following:
And flow over the weir can be calculated by equation:

\[ Q = C_w \sqrt{2g \cdot h^{3/2}} \cdot W \]  \hspace{1cm} (6.3.4)

where \( C_w \) – weir coefficient, which varies with characteristics of weir (for the sharp-crested weir \( C_w = 0.4 \)); \( L \) – length of weir, m; \( W \) – width of weir, m.

### 6.3.3. Required length/diameter

Based on the desired settling velocity of the grit, the length or diameter of tank required to ensure that settling occurs:

For rectangular tanks:

\[ V_H = \frac{Q}{W \cdot h} \]  \hspace{1cm} (6.3.5)

For circular tanks:

\[ V_H = \frac{Q}{3.1416 \cdot D \cdot h} \]  \hspace{1cm} (6.3.6)

And required length - \( L_b \) of grit basin are:

\[ L_b = \frac{h \cdot V_H}{V_s} \]  \hspace{1cm} (6.3.7)

### 6.3.4. Grit storage time

Time available for storage of grit based on the amount of grit produced and the available grit storage volume. The storage of grit in wastewater treatment always are less than 24 hours, due to organic particle, fat and grease fermentation started with unpleasant and dangerous smell for operation personnel in building.
\[ T = \frac{W_{\text{grit}}}{G_{\text{removal}}} \]  

(6.3.8)

where \( V_H \) – flow velocity horizontal component, m/s; \( W \) – width of tank, m; \( D \) – diameter of grit tank, m; \( V_S \) – grit particle sedimentation velocity, m/s; \( Q \) – flow rate, m\(^3\)/s; \( L_b \) – required length of grit basin, m.

6.3.5. **Practical application example - grit chamber design**

**Example 1.** Design a grit chamber of rectangular cross-section. The design Flow (\( Q \)): 100000 m\(^3\); settling velocity of the smallest particle to be removed completely is 0.0236 m/s; Specific Gravity of particles (\( S_s \)): 2.65; horizontal mean flow velocity (\( V \)): 0.30 m/s; Theoretical depth (\( D \)): 2.0 m; calculate the dimensions and slope (mm/m) of the grit chamber. While designing the actual grit chamber, add 25 percent to the depth for grit collection, and 0.25 m freeboard. Also add 25 percent to the theoretically calculated length. Value of Manning’s \( n \) is 0.020. Assume that the volume of grit in the wastewater is 0.10 m\(^3\)/1000 m\(^3\).

**Solution:** Theoretical depth of flow (\( H \)) = 2.0 m

1. Estimate required detention time (\( T \)) = \( H/V_s \) = 2000 / 23.6 = 84.74 s ~ 85 s.
2. Compute theoretical length of channel (\( L \)) = \( V \cdot T = 0.30 \times 85 = 25.50 \) m
3. Flow \( Q = 100000 / 86400 = 1.15 m^3/s \)
4. Compute area of flow (\( A \)) = \( Q/V_s = 1.15 / 0.30 =3.82 m^2 \)
5. Width of the Channel = \( A/H = 3.82 / 2.0 = 1.91 m \sim 2.0 m \)
6. Whetted Perimeter (\( P \)) = \( 2.0 + 2.0 + 2.0 = 6.0 m \)
7. Compute the channel slope \( S \), mm/m:
   
   Using Manning’s equation:
   \[ 0.30 = \frac{1}{0.020} \left( \frac{1.929}{5.0} \right)^{2/3} \times S^{1/2} \]
   
   Using this equation, \( S \) was calculated to be \( 1.12 \times 10^{-4} \)

   Design Length (\( L' \)) = \( 1.25 \times 25.50 = 31.87 m \sim 32 m \)
   
   Design Depth (\( H' \)) = \( 0.35 + 2.0 + 0.25 = 2.6 m \)
   
   Design Width (\( W \)) = 2.0 m
   
   Slope = \( 1.12 \) mm / 32 m
   
   Daily grit production = \( 0.10 \times 100 = 10.0 m^3 \)
6.4. **Clarifiers**

**Objective, principle, size determination**

The sedimentation terms applied to the separation of suspended particles or activated sludge particles that are heavier than water by gravitational settling. The goals of gravity sedimentation are produce a clarified and free of suspended solids wastewater in effluent, second produce a highly concentrated solid sludge stream.

6.4.1. **Sedimentation basin shapes**

They may be rectangular horizontal flow tank, circular radial flow tank and upflow tank and has been presented in Figure 6.4.1. The flow enters at the centre of the tank and settlement takes places as the flow moves outwards and rises, where trough outlet overflow weirs are collected in a channel which conveys the flows to an exit channel.

![Diagram](#)

Figure 6.4.1. Types of sedimentation basins and settling zones there (Tchobanoglous et al., 2003)

In the conventional method (Tchobanoglous et al., 2003) of settling tank design, the smallest settle able particle size under the expected flow rate is selected first, and then corresponding settling (terminal) velocity \( V_s \) is calculated according to the particle physical properties from the Stokes law as (6.3.1), where diameter \( d \) accord to the smallest particle in suspension.

6.4.2. **Classes of sedimentation**

The sedimentation is classified accordingly to the particles settles in clarifiers depending on their physical characteristics and concentration there. Individual particles
may be discrete, such as sand, or flocculent such as activated sludge particles. The particle settling may occur in different flow regimes, such as laminar, transitional or turbulent, and adjustments of velocities calculated from equation (6.3.1), valid for laminar flow, may be required (Tchobanoglous et al., 2003). There are four types of sedimentation classes identified:

**Class 1 of settling**

This is the settling of discrete, non-flocculent particles in a dilute suspension. The particle settles as separate units, and there is no tendency to flocculate or interaction between the particles, like settling of sand in grit chamber.

When a particle settles in a fluid it accelerates until the drag force due to its motion is equal to the submerged weight of the particle. At this point, the particle will have reached its terminal velocity, $V_p$.

A diagram for settling of an idealized spherical particle is shown below (Figure 6.4.2), where $V_p$ – particle settling velocity, m/s; $D$ – drag force; $W$ – submerged weight of the particle; $d$ – diameter of the particle, m; $A_p$ – projected area of the particle normal to the direction of motion, m$^2$; $V_p$ – volume of the particle, m$^3$; $\rho$ – density of the particle, kg/m$^3$; $\rho_p$ – fluid density, kg/m$^3$; $\mu$ – dynamic viscosity of the fluid, N.s/m$^2$; and $C_D$ – drag coefficient.

![Diagram of particle settling](image)

**Figure 6.4.2. Definition diagram for particle terminal settling velocity**

The drag force on a particle is given by:

$$D = C_D \cdot \rho_p A_p \cdot v_s^2 / 2 \quad (6.4.1)$$
In practice, it is found that $C_D$ is a function of the Reynolds Number, $Re$, and, for spherical particles, it can be represented by the following expressions

$$R_e < 1, \quad C_D = \frac{24}{R_e} \quad (laminar \ flow)$$

$$1 < R_e < 10^4, \quad C_D = \frac{24}{R_e} + \frac{3}{(R_e)^{1/2}} + 0.34 \quad (transition)$$

$$R_e > 10^4, \quad C_D \approx 0.4 \quad (turbulent)$$

When the particles settle in laminar regime and this accord to Stokes law and the main design parameter is surface overflow rate – $S=(Q/A)$.

**Class 2 of settling**

This is the settling of suspension with flocculation taking place during the settling process, like sedimentation process in primary clarifier. The flocculated particle velocity increase with time and the design parameters are surface overflow rate, depth of tank or, hydraulic retention time - T. Figure 6.4.3 illustrates the comparison between class 1 and 2 of settling.

![Figure 6.4.3.Comparison between 1 and 2 classes of settling (Tchobanoglous et al., 2003)](image_url)

**Class 3. Hindered settling and zone settling**
This occurs when particles settles as mass and inter-particle forces hinder the settling of the besides particles, so that settlement takes place in a zone. The mass of particles tends to settle as a unit with individual particles remaining in fixed positions with respect to each other. The concentration of mass is intermediate, and will be greater than 1000 mg/l. This usually occurs in secondary clarifier units with the MLSS.

**Class 4. Compression settling**

The concentration is increasing and resulting so high that a structure is formed. The settlement taking place over the resistance provided and occurs through compacting of the structure. Compaction take place through the weight of the particles which is continuously increased by sedimentation from the over lying liquid. This usually occurs in secondary clarifier lower layers of settled sludge mass and activated sludge gravity thickeners.

These settling types are demonstrated in a batch settling test as illustrated in Figure 6.4.4.

![Figure 6.4.4. Settling types in batch test](image)

6.4.3. **Settling basin size determination**

Based on the terminal velocity, the size of the settling tank can be estimated so that all particles with settling velocities equal to or greater than terminal velocity will settle, and particles with settling velocities smaller than either pass through the tank or will be partly removed. The terminal velocity, residence time $T$, and settling tank depth $H$ are related as follows:

$$V_c = \frac{H}{T}$$

\(6.4.2\)
The residence time $T$ can be estimated by the formula:

$$T = \frac{V}{Q}$$

(6.4.3)

where $Q$ and $V$ – inflow rate and tank volume, respectively.

Equation (6.4.2) can be expressed in terms of the surface loading rate ($S$) by substituting Equation (6.4.3) for $T$ and assuming a rectangular tank, for which $V = D \cdot A$, where $A$ – the surface area of the tank and accordingly to (Bretscher et al., 1992):

$$V_c = \frac{D}{V/Q} = \frac{Q}{A} = S_{LR}$$

(6.4.4)

Therefore, the surface area $A$ can be easily obtained from above relationship after terminal velocity and treated flow rate have been determined. However, equation (6.4.4) also implies that the settling tank depth is unimportant in settling tank design, which is obviously not true in practice. The problem appears to be that in the estimation of the $S$, it is assumed that the entire flow passes through the settling tank along the tank surface. However, in the residence time calculation, it assumes that the active flow would occupy the whole tank volume. Thus traditional design methods, which treat the various physical variables as simple averaged parameters without considering the hydrodynamic behaviour of the fluid particle carrier are inadequate for producing an optimal clarifier design.

6.4.4. **Ideal sedimentation basin design practice**

In the engineering design practise there are following assumption make:

1. Settling is class 1 for discrete particles
2. There is an equal distribution of the flow entering and leaving the basin and there also particles distributed equally.
3. Three zones in basin inlet zone, outlet zone and sludge zone.
4. There is uniform distribution of particles through the depth of the inlet zone.
5. Particles move horizontally with the velocity $V$, at the same speed as the wastewater and there is no mixing.
6. Particles that reach sludge zone remain there.
7. Particles that enter inlet zone are removed.
8. The climate factors don’t interrupt the sedimentation process.
The ideal settling basin is given in Figure 6.4.5.

The existing sedimentation tanks work far from ideal, and therefore process models and computer flood dynamic (CFD) software's in design and research are in use.

The use of CFD programs which is applicable to define flow hydraulic patterns for sedimentation tank and inlet structure, see Figures 6.4.6 and 6.4.7.

Figure 6.4.5. Ideal settling basin theory (Row, 1987)

Legend

-0.003220
-0.002505
-0.001783
-0.001061
-0.000384
-0.001106

Vert. velocity, level 13, min = 0.0011 m/s, max = 0.0032 m/s
6.4.5. **Detention Time**

Detention time is defined as the average length of time that flow resides in the clarifier, and is the ratio of the volume of the clarifier divided by the flow rate $Q$:

$$T = \frac{V_{basin}}{Q} \quad (6.4.5)$$

Detention time $T$ is also equal to the basin depth $H$, divided by the settling velocity $V_o$:

$$T = \frac{H}{V_o} \quad (6.4.6)$$

Detention time $T$ is also equal to the basin length $L$, divided by horizontal velocity $V$:

$$T = \frac{L}{V} \quad (6.4.7)$$
Horizontal velocity \( V \) is equal to flow rate \( Q \), divided by cross sectional area \( HB \):

\[
V = \frac{Q}{HB}
\]  

(6.4.8)

Combining equation (6.4.7) and (6.4.8) to eliminate \( V \) gives:

\[
T = \frac{LHB}{Q}
\]  

(6.4.9)

Equations (6.4.6) and (6.4.9) gives:

\[
\frac{LHB}{Q} = \frac{H}{V_o}
\]  

(6.4.10)

Rearranging equation (6.4.10):

\[
V_o = \frac{Q}{LB}
\]  

(6.4.11)

where \( LB \) is clarifier surface area \( A \), and:

\[
V_o = \frac{Q}{LB} = S_{SLR}
\]  

(6.4.12)

6.4.6. **Surface loading rate**

Surface overflow rate is probably the most common and widely used parameter for sizing clarifiers. It’s the ratio of the average flow through the clarifier to the clarifier’s surface area \( m^3/d \): 

\[
S_{LR} = \frac{Q}{A}
\]  

(6.4.13)

6.4.7. **Solids loading rate**

Solids loading rate takes into consideration the concentration or weight of solids that must settled by the clarifier. Solids loading rate is expressed as the weight of solids per unit of clarifier surface area per unit time, usually per day:

\[
SSLR = \frac{(TSS \times Q)}{A}
\]  

(6.4.14)
6.4.8. **Weir loading rate**

The weir loading rate is the ratio of the average flow through the tank to the weir length in the clarifier:

\[ W_{LR} = \frac{Q}{\text{Weir Length}} \]  

(6.4.15)

6.4.9. **Surface area**

The required surface area for a clarifier can be based upon the solids loading to the clarifier and the chosen solids loading rate, or:

\[ A = \frac{S_{LR}}{\text{Recommended } S_{LR}} \]  

(6.4.16)

The recommended solids loading rate depends on type of clarifier and are given in (6.4.16), where \( A \) – surface area, m\(^2\); \( Q \) – flow rate, m\(^3\)/s; \( V \) – volume, m\(^3\); \( TSS \) – total suspended solids, mg/l.

6.4.10. **Practical application example -sedimentation tank design**

**Example 1.** Design a rectangular sedimentation tank to treat 10000 m\(^3\) of raw wastewater per day. The detention time may be assumed to be 2 hours.

**Solution:**

1. Estimate the volume of tank = Flow x Detention time = 10000 x 2/24 = 800 m\(^3\)
2. Assume depth of tank = 3.0 m.
3. Compute tank surface area = 800/3 = 267 m\(^2\)
4. Assume the length and width L/B = 2 and L = 2B.
5. Compute B and L:
   \[ 2B^2 = 267 \text{ m}^2 \text{ i.e. } B = 11.55 \text{ m} \]
   \[ L = 2B = 11.55 \times 2 = 23.10 \text{ m} \]
6. Compute the actual surface loading, which must be less than given standard for rectangular clarifier.
   Hence surface loading (Overflow rate) = 37,453 l/d/m\(^2\) < 40,000 l/d/m\(^2\)
6.5. Trickling filters

The trickling filter is a biological attached grow wastewater treatment unit in which microorganisms responsible for treatment are attached to an inert packing material. A trickling filter bed consists of coarse materials, such as stones, gravel, and a wide range of plastic media. The wastewater in trickling filter is distributed over the top area of a filter bed containing non-submerged packing material and trickles down getting biofilm treatment in the process. Air circulation in the void space, by either natural draft or blowers, provides oxygen for the microorganisms growing as an attached biofilm. During operation, the organic material present in the wastewater is metabolised by the biomass attached to the medium. The biological slime grows in thickness as the organic matter abstracted from the flowing wastewater is synthesized into new cellular material.

The oxygen, organic matter and nutrients are transferred to the fixed film by diffusion. The oxidation of organic matter are transferred to moving after layers by diffusion and due to bacteria metabolism of the fixed film grows and gets thickened. As it’s thickness increases, inner layer become anaerobic. The outer layer remains aerobic with more thickening of biological layer, cells in inner layer die and lyses It causes break in the content between film and medias. Thus, slim layer gets slough off and is being carried.

Figure 6.5.1. Trickling filter (Tchobanoglous et al., 2003)
away by the wastewater to separate device like secondary clarifier. The trickling filter treatment processes are given in Figure 6.5.2.

![Diagram of trickling filter process](image)

Figure 6.5.2. Schematic diagram of attached growth process (Tchobanoglous et al., 2003)

### 6.5.1. Hydraulic design equations

The design of trickling filters is based on empirical and mass balance concepts. The equations have been developed for calculating removal efficiency of BOD on the basis of factors such as bed depth, media characteristics, recycling rate, temperature and loading rates. The design is based on NRC (National Research Council of USA) and Eckenfelder equations (Lawrence et al., 2009), which general are used in practise.

#### 6.5.1.1. NRC Formula:

\[
\frac{S_O - S_e}{S_O} = \frac{1}{1 + 0.532 \cdot \sqrt{\frac{QS_o}{VF}}} \quad (6.5.1)
\]

\[
F = \frac{1 + r}{(1 + 0.1 \cdot r)^2} \quad (6.5.2)
\]

where \(S_O\) – Incoming BOD\(_5\), mg/l; \(S_e\) – effluent BOD\(_5\), mg/l; \(Q\) – flow rate, m\(^3\)/h; \(V\) – volume of filter media, m\(^3\); \(F\) – recirculation factor; \(R\) – recycling rate.

#### 6.5.1.2. Eckenfelder Formula

Formula expresses account for media characteristics and valuable as first order kinetics.

No recycling:


\[
\frac{S_e}{S_o} = e^{-KD/Q^n} \tag{6.5.3}
\]

With recycling:

\[
\frac{S_e}{S_o} = \frac{e^{-KD/Q^n}}{(1+R) - Re^{-KD/Q^n}} \tag{6.5.4}
\]

where \(K\) – treatability constant, \(\text{min}^{-1}(0.01 – 0.1)\); \(D\) – depth, \(\text{m}\) (Domestic – 0.06 at 20°C); \(N\) – coefficient, characteristic of medium = 0.35… 0.80 (rock = 0.408 asbestos = 0.75); \(Q\) – \(\text{m}^3/\text{m}^2\text{min}\); \(K_{20}\) – temperature coefficient for 20°C. For actual temperature \(K_T\) is calculated:

\[
K_T = K_{20} \cdot (1.035)^{T-20} \tag{6.5.5}
\]

6.5.1.3. Hydraulic Loading

The hydraulic loading \(Q_{OVR}\) is ratio of flow through the trickling filter to the trickling filter surface area:

\[
Q_{OVR} = Q / A \tag{6.5.6}
\]

6.5.1.4. Organic Loading

The organic loading \(Q_{BOD}\) rate of \(BOD\) loading per volume of trickling filter media:

\[
Q_{BOD} = (BOD \times Q) / V \tag{6.5.7}
\]

6.5.1.5. Flushing Intensity

The amount of flow distributed to the trickling filters by the trickling filter arms per revolution:

\[
\text{Flushing intensity} = \text{Hydraulic loading} / (\text{No. of arms} \times \text{Distributor speed}) \tag{6.5.8}
\]

6.5.1.6. Distributor Rotational Speed

The amount of time it takes the distributor to make one full pass around the trickling filter.
6.5.1.7. Actual velocity

The actual velocity of wastewater particles trickling through a medium is so called seepage velocity. For practical use the water can be assumed to flow through the medium with the so called discharge velocity \( V \) and calculated by formula:

\[
V = \frac{Q}{A \times t} = \frac{q}{A}
\]

(6.5.10)

where \( V \) – effluent velocity, m/s; \( Q \) – treated wastewater, m\(^3\); \( A \) – cross section area, m\(^2\); \( T \) – time, s; \( q \) – flow rate, m\(^3\)/s.

6.5.1.8. Pressure head

There hydraulic gradient \( I \) is the ratio of head loss flow distance between two points and is calculating by formula:

\[
I = \frac{h_1 - h_2}{L} = \frac{\Delta h}{L}
\]

(6.5.11)

where \( h_1 \) – hydraulic head in point upstream, mm; \( h_2 \) – hydraulic head in point downstream, mm; \( \Delta h \) – difference in hydraulic head between points upstream and downstream, m; \( L \) – distance between the points, mm.

For a filter bed column the hydraulic gradient describes the ratio of total head either side of the filter to the thickness of the bed direction of the flow.

Accordingly to Darcy Law the flow velocity \( V_d \) through filter media is directly proportional to the hydraulic gradient as follows:

\[
V_d = -\frac{K \cdot H_L}{L}
\]

(6.5.12)

where \( V_d \) – average flow velocity, m/s; \( K \) – hydraulic conductivity, m/s; \( H_L \) – head loss; \( L \) – length of medium, m.
6.5.2. Practical application example - trickling filter design

**Example 1.** Design a low rate biofilter to treat 6000 m$^3$ of sewage of BOD$_5$ of 210 mg/l. The final effluent should be 30 mg/l and organic loading rate is 320 g/m$^3$/d.

**Solution:** assume 30% of BOD load removed in primary sedimentation i.e., = 210 x 0.30 = 63 mg/l. Remaining BOD = 210 - 63 = 147 mg/l. Percent of BOD removal required = (147-30) x 100/147 = 80%.

BOD load applied to the filter = flow x conc. of sewage (kg/d) = 6 x 10$^6$ x 147/10$^6$ = 882 kg/d.

To find out filter volume, using NRC equation without recycling, where F=1:

$$\frac{S_o - S_e}{S_o} = \frac{1}{1 + 0.532 \cdot \sqrt{\frac{QS_o}{VF}}}$$

$$E = \frac{S_o - S_e}{S_o}, \text{ where } E \text{ - treatment efficiency, } E = 80\%$$

$$80 = \frac{100}{1 + 0.532\left(\frac{882}{V\times 1}\right)^{1/2}}$$

$$V = 2704 \text{ m}^3$$

Depth of filter = 1.5 m, Filter area = 2704/1.5 = 1802.66 m$^2$, and Diameter = 48 m < 60 m.

Hydraulic loading rate = 6 x 10$^6$/10$^3$ x 1/1802.66 = 3.33 m$^3$/d/m$^2$ < 4 (according to standard)

Organic loading rate = 882 x 1000 / 2704 = 326.18 g/d/m$^3$ which is approx. equal to 320.

6.6. Activated sludge

The hydraulic behaviour in an activated sludge tank, i.e., the transportation of MLSS (mix liquor suspended solids) through the tank, is of fundamental importance for the function of the wastewater treatment process and therefore for efficiency. Many examples of hydraulic phenomena with negative effect on sludge tank performance include short circuiting streams and dead volumes. These unfavourable hydraulic conditions in activated sludge tanks may cause the less process efficiency and thus can obtain different from design biological kinetic rates, different grow rates and yield coefficients. In continuously
operated tanks there are two main types of theoretical hydraulic behaviour, plug flow and complete mix.

6.6.1. Hydraulic characteristics of reactor tanks

Mixing regimes in reactor tanks may be plug flow or completely mixed system and illustrate the Figure 6.6.1.

![Illustrations of plug-flow (a) and complete-mix (b) hydraulic behaviour](Tchobanoglous et al., 2003)

6.6.1.1. Plug flow system

- Each element has the same residence time;
- Long and narrow in dimension;
- No longitudinal mixing;
- BOD highest in inlet and DO lowest at inlet;
- Theoretically more efficient than completely mixed flows.

6.6.1.2. Completely – mixed system

- Each element may not have the same HRT (hydraulic resident time);
- Continuous and thorough mixing;
- Rectangular tanks typically 6-7 m width and 3-5 m depth;
- Uniform MLSS and BOD;
- Substrate theoretical concentration in tank and effluent are equal;
- Better resistance to shock hydraulic and pollutants loads.

Ideal plug-flow is characterized by fluid particles passing through the tank and being discharged in the same sequence as in which they entered. The particles remain in the tank for a time equal to the theoretical detention time. This type of flow is approximated in long
tanks with a high length-to-width ratio (Figure 6.6.1.a). These types of reactors are also known as tubular flow reactors (Tchobanoglous et al., 2003).

Ideal complete-mix occurs when the fluid particles entering the tank are immediately dispersed throughout the tank. There are no concentration gradients in the tank and the composition is equal all over the tank. Therefore, the effluent of the tank has the same composition as the fluid inside the tank. This type of flow is approximated in round or square tanks if the content of the tank is uniformly and continuously redistributed (Figure 6.6.1.b). Complete-mix reactors are also known as continuously-stirred-tank-reactors, CSTRs (Tchobanoglous et al., 2003).

The actual hydraulic behaviour of most activated sludge tanks treating wastewater, fall somewhere in-between plug-flow and complete-mix. It is therefore necessary to characterise the hydraulic behaviour within an individual tank, if its effects on the treatment process is to be understood (Burrows et al., 1999).

6.6.2. **Hydraulic phenomena in activated sludge tanks**

The hydraulic behaviour in an activated sludge tank, i.e. the transportation of mixed liquor (MLSS) flow through the tank, is of fundamental importance for the function of tank and thus for efficiency of the activated sludge treatment process. Good hydraulic conditions are characterised by good conditions for appropriate biochemical reaction kinetics and grow rates for desirable microorganisms.

In real reactors various hydraulic phenomena may occur. Two common phenomena are short circuiting streams and dead volume. A short circuiting stream is a situation in the tank where the incoming flow or a part of the incoming flow is taking a short cut, thus bypassing the reactor (Figure 6.6.2). A short circuiting stream will therefore have a low residence time in the tank. Dead volumes (or dead zones) are water volumes in the tank that are stagnant. Typically, the volume near a corner in a tank may act as dead volume if the mixing is insufficient (Figure 6.6.2). In dead volumes there are none (or little) exchange between the bulk flow in the tank and the dead volume. Dead zones reduce the effective reactor volume; as a consequence, the active reactor volume is smaller than expected, see Figure 6.6.3.
The performance of an activated sludge process is influenced by its hydraulic behaviour. The hydraulic behaviour is in its turn affected by a number of factors such as; the geometric design of the reactor, the shape and position of the inlet and the outlet, external mixers, baffles, fluid viscosity, aeration and water flow rate. An unfavourable
hydraulic situation in an activated sludge tank may lead to significant reduction of its capacity, thus causing higher concentration of residuals in the effluent.

The mixing will also have a pronounced effect on the concentration of substrate available to the microorganisms and this will, in turn, affect the population of microorganisms present (Tchobanoglous et al., 2003). Sludge bulking is a term applied to a condition in which an overabundance of filamentous organisms is present. The filamentous organisms cause the biological flocs in the reactor to become bulky and loosely packed. Bulky flocks do not settle well and are often carried over in great quantities in the effluent of the sedimentation tank.

### 6.6.3. Aeration

The most common application of gas transfer in the field of wastewater treatment is two film theories – physical mass transport across gas film and liquid film. For the transfer of gas molecules from the gas phase to the liquid phase, slightly soluble gases encounter the primary resistance from the liquid film. Very soluble gases encounter to transfer from the gaseous film. To transfer large quantities of oxygen that are needed, different aeration devices are used. The submerged diffused air devices such as porous plates or tubes from coarse bubble till fine bubbles diffusers are in use. There also surface aerators and rotor brush aerators are applicable. The aim of aerator is to increase oxygen transfer from liquid film to the bulk liquid at a rate sufficiency to meet the oxygen demands of metabolism. In the wastewater treatments plants the aeration systems is the major energy consuming process. The oxygen transfer rate from air bubbles into solution is written as (Hammer, 1986):

\[
 r = K(\beta \cdot C_S - C_t)
 \]  

(6.6.1)

where \( r \) – rate of oxygen transfer from air to liquid, mg/l h; \( K \) – transfer coefficient, per h; \( \beta \) – oxygen saturation coefficient of the wastewater (often 0.8-0.9); \( C_S \) – DO concentration at saturation of top water, mg/l; \( C_t \) – DO concentration in mixed liquor, mg/l; \( \beta C_S - C_t = DO \) deficit, mg/l.

The greater the \( DO \) deficit, the higher the rate of oxygen transfer. The oxygen requirements depend in general on maximum organic loading rate \( BOD \), degree of treatment, and \( MLSS \). The normal design recommends for activated sludge processes except extended aeration about 80 - 90 m\(^3\)/kg \( BOD_5 \).
6.6.3.1. Air piping

As compressed air flows through a pipe, its volume changes according to the pressure drop. The pressure drop $h_f$ in a pipeline can be calculated by the Darcy-Weisbach formula:

$$h_f = f \cdot \frac{L}{D} \cdot \frac{V^2}{2g}$$ \hspace{1cm} (6.6.2)

where $h_f$ – head or friction losses, cm; $f$ – friction factor; $L$ – length of pipeline, cm; $D$ – diameter of pipe, cm; $V^2/2g$ – velocity head, cm; $V$ – velocity of flow, cm/s.

The value of friction factor $f$ can be determinate from the Moody diagram. However, as an approximation, the friction factor of a steel pipe is expressed as (Steel and McGhee, 1979):

$$f = \frac{0.029 \cdot D^{0.027}}{Q^{0.148}}$$ \hspace{1cm} (6.6.3)

where $f$ – friction factor; $D$ – diameter of pipe, m; $Q$ – flow of air, m$^3$/min.

The headloss in a straight pipe is calculated from equation:

$$H = f \cdot \frac{L}{D} \cdot h_r$$ \hspace{1cm} (6.6.4)

where $H$ – headloss in pipe, mm; $L$ – length of pipe, m; $D$ – diameter of pipe, m; $h_r$ – velocity head, mm.

6.6.4. Design aspects

In the activated sludge process, the quality of the effluent is to a large extent dependent on how the biological sludge mass can be separated from the treated wastewater. The solids/liquid separation traditionally take place through a separation of spontaneously aggregated flocks of activated sludge by gravity sedimentation in secondary clarifiers.

Typical designs for activated sludge systems do not ease the effects of hydraulic over loadings. However, the pilot studies conducted by (Olsson, 1986) showed that the provision of freeboard in the aeration basin coupled with the use of an appropriate effluent weir design lessen the hydraulic effect and reduced the effluent suspended solids.
6.6.4.1. Required aeration volume

The required aeration volume is the volume of the aeration tank required based on flow, BOD loading, and recommended BOD loading rate:

\[ V_{\text{required}} = \frac{(\text{BOD loading} \times Q)}{\text{Loading factor}} \]  
(6.6.5)

6.6.4.2. Return activated sludge (RAS) rate

The return activated sludge rate is the rate of flow that should be returned from the secondary process back to the activated sludge system to maintain the optimum microbial mass within the activated sludge process:

\[ \text{RAS rate} = Q \times \left( \frac{\text{MLSS}}{\text{RAS} - \text{MLSS}} \right) \]  
(6.6.6)

6.6.4.3. Solids retention time (SRT)

Solids retention time is the average length of time that the solids mass (microorganisms) remain the activated sludge and secondary system (including return flow):

\[ \text{SRT} = \frac{\text{MLSS} \times V}{\text{(WAS} \times Q_{\text{waste}}) + (\text{ESS} \times Q)} \]  
(6.6.7)

6.6.4.4. Food to mass (F/M) ratio

The ratio of food (soluble BOD) to mass (microorganisms – MLSS or MLVSS) in the activated sludge system:

\[ \text{F/M Ratio} = \frac{\text{BOD} \times Q}{\text{MLVSS} \times V} \]  
(6.6.8)

6.6.4.5. Sludge wasting rate, mass based

Sludge wasting rate is the rate of excess sludge flow that should be removed from the activated sludge system:

\[ \text{Sludge wasting rate} = \frac{(\text{Aeration MLVSS} \times V)}{(\text{SRT} \times \text{RAS MLVSS})} \]  
(6.6.9)

Sludge wasting rate is the rate of excess sludge flow that should be removed from the activated sludge system. Note that RAS rate below is the percentage of RAS. For example, if the RAS rate is 50%, then 0.5 should be used in the calculation:

\[ \text{Sludge wasting rate} = \frac{\text{RAS rate} / (1 + \text{RAS rate})}{(V / \text{SRT})} \]  
(6.6.10)
6.6.4.6. Oxygen requirements

The amount of oxygen required for the microorganisms to metabolize the organics in the wastewater, typically expressed in kg/day:

$$O_2\text{required} = (BOD - O_2\text{required} \cdot BOD \cdot Q) + (NH_3 - O_2\text{required} \cdot NH_3 \cdot Q) \quad (6.6.11)$$

where $Q$ – flow rate; $V$ – volume; MLSS – mixed liquor suspended solids; MLVSS – mixed liquor volatile suspended solids; $BOD$ – biochemical oxygen demand; $RAS$ – return activated sludge; WAS – waste activated sludge; $SRT$ – solids retention time; $ESS$ – effluent suspended solids; $F/M$ – food to mass ratio.

6.7. Sludge Thickeners

Thickening and dewatering are important component of sludge management programs. Thickening technologies include gravity, centrifugal, gravity belt, and rotary drum thickeners. These thickening methods differ significantly. The following section describes the process and hydraulic behaviour in gravity thickeners.

In gravity thickening, solids are concentrated by the gravity induced settling and compaction. Gravity thickeners are used for primary and waste activated sludge or both combination thickening. Primary solids from thicken easily as they settles quickly and achieve solids concentration 4-6%, but waste activated sludge settle slowly and resist compaction, resulting loading rates, apart from that have a tendency to stratify because of the flotation effect of gas produced from continuing biological activity.

Gravity thickener typically consist of circular tank with a side water depth of 4 till 6 m. The diameter various significantly depending on hydraulic loading rate and detention time, also from construction requirements, see Figure 6.7.1.
6.7.1. **Hydraulic loading rate**

The ratio of the influent sludge flow to the thickener surface area:

\[
\text{Hydraulic loading} = \frac{Q}{\text{Area}} \tag{6.7.1}
\]

6.7.1. **Sludge detention time**

The amount of time that sludge is in the thickener before it is removed for further processing:

\[
\text{Detention time} = \frac{V}{Q} \tag{6.7.2}
\]

6.7.2. **Solids loading rate**

The ratio of the amount of solids entering the thickener over a given period of time to the thickener’s surface area:

\[
\text{Solids loading} = \text{Influent TSS} \times \frac{Q}{\text{Area}} \tag{6.7.3}
\]

where \( V \) – volume; \( Q \) – flow; \( TSS \) – total suspended solids.
6.8. Wastewater treatment plant hydraulic profiles

Hydraulic profile is the graphical representation of the hydraulic grade line through the wastewater treatment plant, see Figure 6.8.1. As usually, the highest water level in of the receiving water is known, this level is used as a hydraulic profile start point, and the head loss computations are there started backward through the plant till effluent in the water body, as the rules the pass flow the treatment units by gravity. The total available head at the treatment plant is the difference in water surface elevations in the inlet chamber and the surface elevation in the receiving water at highest peak flow. And if the total head loss is less than the head losses through the plant, flow by the gravity cannot be achieved. In such cases pumping is needed to rise up the head so that flow by gravity can occur.

Figure 6.8.1. Waste water treatment plant hydraulic profile

The hydraulic profiles are prepared at peak and average design flows and at minimum initial flow, and are prepared for all main paths and process units though the plant. The head loss though the treatment plant is the sum of head losses in the treatment units and the connecting piping or channels. The head losses include the head losses of the influent and effluent structures, and losses through the all process units. And the same for unit connecting piping and channels, also there head losses due to bends, fittings, gates, valves, and meters are taking into consideration.
7. Gas and liquid streams diagnosis

Nina Karbalevich
Belarusian State University, Physical Faculty
7.1. Flow sensors

In the investigation of heat transfer processes of the bodies’ surfaces with the environment, the speed of the flow is an important characteristic that affects the heat transfer rate. Devices intended for investigation liquid or gas flows are called flow sensors. Talking about the flow rate, we mean scalar value (i.e. the modulus of the velocity vector), since most discussed in this chapter flow meters don’t provide information about the direction of flow. The term «fluid» may refer to a liquid or gas.

Different flow meters provide measuring of different flow characteristics. Electromagnetic and ultrasonic sensors intended for measuring mean flow velocity. Thermal and hydrodynamic (using a Pitot tube) methods allow measuring the rate of the local environment in the stream.

7.1.1. Thermal flow meters

In heat flow meters heating element mounted in the flow path of liquid or gas and streamlined by this stream is used. Heat is transferred from the element to the fluid with the intensity $P$, depending on the temperature difference between the element and the fluid, its specific heat capacity $c$ and rate $\nu$, and also the flow profile. Convectional rate measurement method based on the principle of heat transfer from the heating element into the flow.

Convectional flow meter provides determination of the local speed of the fluid or gas by measuring the amount of heat which dissipates by the heated element, streamlined by the flow. Any resistive heater can be used as such an element: tungsten wire, a thermistor or a metal film on the substrate. The element is heated above the temperature of the fluid by passing therethrough an electric current (Joule heating).

Dissipated power is approximately given by the King law in this case:

$$P = S \left( T_E - T_M \right) C_0 + C_1 \nu^{1/2}, \quad (7.1.1)$$

where $P$— heat emission power; $S$— the effective area of the heating element; $T_E$— element temperature; $T_M$— medium temperature; $\nu$— flow rate; $C_0$, $C_1$— empirical coefficients determined by the structure of the heating element and the specific heat capacity of the
fluid. The heated element and the fluid are in thermal equilibrium, the amount of heat transferred into the flow in each second is equal to the Joule power allocated in the element:

\[ P = I^2R, \tag{7.1.2} \]

where \( P \)– power allocated in the element; \( I \)– the electric current through the element; \( R \)– electrical resistance of the element. From the relations (7.1.1) and (7.1.2) it follows:

\[ I^2 = \frac{S(T_E - T_M)(C_0 + C_1\sqrt{\nu})}{R} \tag{7.1.3} \]

Equation (7.1.3) contains four variables: \( CI, T_E, T_M \) and \( \nu \). Since the resistance \( R \) of any resistive element is connected with its temperature \( T_E \), then from this equation the flow rate at a constant \( I \) or \( R \) can be determined. If a constant amount of current \( I \) is maintained, then the element temperature will vary widely under the liquid or gas velocity changing, which leads to unsatisfactory sensitivity of the measuring devices at high speeds and the possibility of thermal damage (burning) of a resistive element at low flow velocities. Therefore, the value of resistance \( R \) usually held constant, while the resistance element temperature is obviously constant. Knowing the fluid medium temperature \( T_M \), the rate can be found from equation (7.1.3) as a function of the current \( I \).

The feedback system needed to maintain a constant value of \( R \) in the flow meter with the fixation of the resistive element temperature provides significantly higher performance of the device compared with the measuring instrument, wherein the current value is fixed.

The probe with the heated element may have various designs. The wire probe (Fig. 7.1.1a) most often used to measure the air velocity. To ensure an adequate sensitivity, a thin tungsten wire of the probe is heating up almost red hot. In Fig. 7.1.1b a thin film sensor is used for measuring the supersonic velocities in the air is shown. A thin platinum film deposited on an insulating substrate, and the result is a very durable structure. Both wire and thin-film sensors are characterized by a positive temperature coefficient of resistance (TCR), i.e. when the temperature increases their resistance increases too.

To ensure the necessary operating mode of the probe (with constant temperature), the temperature control circuit with the feedback is needed.

It is important to calibrate the measuring system for accurate measurements. Since the scheme can be paired with a computer, the conversion of the measured value of the current \( I \) in the flow rate \( \nu \) can be performed using a memorized computer conversion table.
Fig. 7.1.1. The wire probe (a) and a thin film sensor (b)

Devices with a thermal resistance used to measure the flow rate of the gas, are called thermoanemometers. Thermoanemometer is the primary means of measuring the characteristics of turbulent flows, especially fluctuating velocity components. This becomes possible due to the low thermal inertia of the measuring thread.

Thermoanemometer scheme is shown in Fig. 7.1.2. The sensing element (sensor) of the thermoanemometer is a thin platinum or tungsten filament diameter of 5 µm and 1 mm long, stretched between needle-holders. The filament is included in the circuit as a resistance thermometer and heated by direct electric current.

Fig. 7.1.2. Thermoanemometer scheme

Since intensity of heat exchange between the heated filament and the gas flow depends on the rate of the latter, the filament temperature will vary depending on the rate change.

In practice, the heat losses by convection $Q$ are usually calculated by Newton formula:
\[ Q = \alpha S (T_w - T_M), \]  
(7.1.4)

where \( \alpha \) – heat transfer coefficient; \( S \) – the surface area of the wire immersed in liquid or gaseous medium; \( T_w \) and \( T_M \) – wire temperature and temperature of the medium.

The thermal equilibrium equation for the thermoanemometer transducer has the form:
\[ I^2R = \alpha S (T_w - T_M) = \alpha S \theta, \]  
(7.1.5)

where \( \theta = T_w - T_M \).

The task of constructing the transducer (thermal resistance) is to reduce as far as possible to zero all the heat losses by the wire, except the heat loss by convection. In the thermal equilibrium equation there are no terms that express the heat losses due to the wire heat conduction and due to radiative heat transfer.

In the thermoanemometers converters ratio of the wire length to its diameter is up to 1000 or higher, which allows us to neglect heat loss by the wire heat conduction. As for the radiation losses, then when measuring, for example, the air velocity in open flow, these losses are very small; in addition, they are considered by grading the tool.

The heat transfer coefficient can be calculated from equation (7.1.5).

\( \alpha \) value depends on many variables, including the medium motion speed \( \nu \), as well as on the configuration of transmitter.

The functional dependence of these quantities is determined by using the theory of similarity. With regard to physical phenomena the concept of similarity is applicable only to the phenomena of the same kind, described by a set of the same equations.

In the analysis of such phenomena it is possible to compare with each other only similar quantities and only at similar points in space and at similar points in time.

The link between Reynolds and Nusselt criterions is proved in the theory of similarity:
\[ Nu = f \text{ Re}. \]  
(7.1.6)

Reynolds criterion \((Re)\) binds the velocity of the medium and its viscosity with geometrical parameters characterizing the motion of the medium:
\[ Re = \nu L \text{ } \nu = \nu p L \eta. \]  
(7.1.7)
where $\eta$ – dynamic viscosity; $\nu$ – kinematic viscosity; $\rho$ – medium density; $l$ – the determining geometric size.

Diameter $d$ is the determining size at medium movement in a cylindrical tube or under flow around the cylinder.

Nusselt number (Nu) is derived from the equation that defines the conditions of heat transfer between flow and a solid surface:

$$Nu = \frac{\alpha d}{\lambda},$$

where $d$ – the determining geometric size (diameter of the cylinder); $\alpha$ – heat transfer coefficient; $\lambda$ – coefficient of medium thermal conductivity.

Dependencies between similarity criteria are usually presented in the form of power functions. For example, for the same atomic gases in the outer flow around cylindrical bodies:

$$Nu = cRe^m,$$

where $c$ and $m$ – constants.

The values of $c$ and $m$ constants from the relationship (7.1.9) are given in Table 7.1.1 for different ranges of $Re$, obtained at the investigation of cylinders with diameter from 0.099 to 155 mm heat transfer at an air flow from 2 to 30 m/s.

<table>
<thead>
<tr>
<th>$Re$</th>
<th>$c$</th>
<th>$m$</th>
</tr>
</thead>
<tbody>
<tr>
<td>5 – 80</td>
<td>0.81</td>
<td>0.40</td>
</tr>
<tr>
<td>80 – 5·10³</td>
<td>0.625</td>
<td>0.46</td>
</tr>
<tr>
<td>≥5·10³</td>
<td>0.197</td>
<td>0.6</td>
</tr>
</tbody>
</table>

For thermoanemometers transducers from a very thin wires (with diameter 0.015 – 0.03 mm) at flow rates up to 60 m/s $Re$ criterion value doesn’t exceed 30. Therefore, the following values: $m = 0.4$, $c = 0.81$, should be chosen to calculate the flow rate.

Thermoanemometer can be included in a bridge circuit (Fig. 7.1.3). The measurements can be performed by maintaining a constant current in either a straight section of the bridge or the bridge supply voltage (then the bridge will work in a non-equilibrium mode), or the value of the thermoanemometer resistance $R$ (then the bridge will operate in the balance, as each speed values need to install the appropriate current value).
Fig. 7.1.3. Thermoanemometer measuring circuit

For measurements in a nonstationary mode can be used only non-equilibrium bridge, i.e. measurements at $I = \text{const}$.

The example of a thermoanemometer calibration curve $R=f(\nu)$ is shown in Fig. 7.1.4 at $I = \text{const}$.

As can be seen from the figure, the scale is distinguished by significant non-linearities. Another large scale non-linearity will occur when using the equilibrium bridge circuit (thermoanemometer resistance is kept constant), since the current is needed be changed proportional to about the fourth root of the measured speed (at $m = 0.35 - 0.6$).

Fig. 7.1.4. Thermoanemometer calibration curve

To increase the scale of the character in this case it is necessary to measure the voltage drop on a platinum wire instead of the current (Fig. 7.1.2). Since the value of $R$ increases with increasing temperature, i.e. with increasing I, the dependence $IR=f(\nu)$ will be more linear than the dependence $I=f(\nu)$. 
Typically sensors are calibrated using the specific calibration means, such as the measuring nozzle.

Working wire temperature heated by current when it is used as a thermoanemometer is chosen sufficiently high, of the order of 600 – 800 °C. When the wire temperature is higher, the sensitivity of the device becomes higher as a whole. Increase of the wire temperature is limited by considerations of its properties stability. If the wire is made of base metals such as copper or constantan, then at a temperature above 600 – 700 °C (and for copper – above the 200 °C) there is a risk of the wire oxidation. Platinum wire can be heated to higher temperatures, but almost above the 700 – 800 °C it is not heated in practice.

![Diagram](image)

**Fig. 7.1.5.** Dependence of the relative heat transfer of the wire heated by current from the angle between the wire and the direction of the air flow

The thermoanemometer wire must be disposed, if it is possible, perpendicular to the direction of the flow. The curve describing the dependence of the wire heat transfer coefficient on the angle between the wire and the direction of the flow is shown in Fig. 7.1.5. The curve shows that the 10° change in the wire slope from perpendicular practically does not cause an error (so the wire is mounted perpendicular to the flow). Further angle change causes a very significant change in thermoanemometer indications, which is used for measuring the direction of the flow.

### 7.1.2. Hydrodynamic (aerodynamic) flow meters

The communication between a pressure difference at the two characteristic points of the flow and flow rate is used in measuring devices of this type. This pressure difference is measured using a differential pressure transducer connected to a computer.

Pitot tube (Fig. 7.1.6) provides a measure of the local rates in the fluid or gas flow.
There are two types of holes, opened for fluid flow, in the tube. The static fluid pressure is operating on static inlet, formed as an opening whose axis is perpendicular to the direction of flow. Input (hole) at the end of the tube collects braked mass of liquid or gas which is under full pressure.

If the Pitot tube is parallel to the flow, then the difference between these two pressures can be found from Bernoulli equation:

\[ \Delta p = p_0 - p = \frac{\rho v^2}{2}, \]  

where \( \Delta p \) – pressure difference; \( p \) – pressure on static inlet; \( p_0 \) – full pressure; \( \rho \) – density of the fluid; \( v \) – flow rate. Usually inputs of total and static pressure are connected to a differential pressure gauge and the dynamic pressure \( \Delta p = p_0 - p \) is measured directly. Flow rate of incompressible fluid is determined by the formula:

\[ v = \frac{2 \ p_0 - p}{\rho}^{1/2} \]  

This ratio does not take into account the influence of the viscosity and therefore it is valid only at high Reynolds numbers.

For a compressible fluid (e.g., air) flow rate is determined by the formula:
$\nu = \frac{2k}{k-1} \frac{\rho}{p_0} \frac{1}{\frac{k}{k-1}} - 1$ \hspace{1cm} (7.1.12)

where $k = c_p / c_v$ – the ratio of specific heat capacities.

Pitot tube is commonly used to measure the air velocity in the chimney and aircraft and can be used in any fluid medium. This is a very accurate device that requires minimal maintenance. The main drawback of Pitot tube is a low sensitivity at low flow rates, and the nonlinear dependence of the rate on the pressure difference.

Measurements of the total and static pressure can be carried out by one sensor – Prandtl tube.

![Fig. 7.1.7 Prandtl tube](image)

### 7.1.3. Electromagnetic flow meters

Electromagnetic flow meters are used to measure the flow velocity averaged over its cross-section, and are suitable for almost all conductive liquids. Their work is based on the phenomenon of electromagnetic induction: in any conductor moving perpendicular to the direction of the magnetic field the EMF is induced. The EMF magnitude is directly proportional to the velocity of the conductor (in this case – the velocity of the fluid).

The principle of the electromagnetic flow meter operation is explained in Fig. 7.1.8. The magnetic field of induction $B$ is applied perpendicular to the direction of liquid flow. The induced electric field determining the electromagnetic induction EMF is perpendicular both to the direction of flow and to the direction of the magnetic field. EMF is determined by Faraday law:
\[ \varepsilon = \int_{a}^{b} \nu \times B \cdot dL, \]  

(7.1.13)

where \( \varepsilon \) – the EMF induced between points \( a \) and \( b \); \( B \) – magnetic induction; \( a, b \) – points at which the electrodes are; \( L \) – the distance between points \( a \) and \( b \); \( \nu \) – liquid flow rate.

Fig. 7.1.8 Electromagnetic flow meter

Assuming that the spatial distributions of the induction \( B \) and the rate \( \nu \) are uniform and the vectors \( B, \nu \) and \( L \) are orthogonal, we obtain the following expression for the EMF:

\[ \varepsilon = BL\nu. \]  

(7.1.14)

The induced EMF is measured by two electrodes contacting with the liquid. The type of electrodes used depends on the type of liquid. The electrodes made from non-reactive metals, such as platinum, are used for many fluids, including liquid metals. The relations (7.1.13) and (7.1.14) are valid only for the axially symmetric flow. It is the case that often occurs in practice. For a stream with a known asymmetric profile, a correction factor would be entered in the flow meter indications.

Between electrodes, which are in liquid, there may be the potential difference (as in a conventional electrochemical cell), which is manifested in the presence of a bias voltage of zero for the measured EMF. This problem doesn’t occur when using an alternating magnetic field. The electrodes may be exposed to corrosion; they are gradually deposited by foreign materials, which affect their resistance and the measured EMF.

The inhomogeneity of the magnetic field along or perpendicular the flow axis can lead to significant errors due to the formation of local currents circulating in the liquid.
7.1.4. **Ultrasonic flow sensors**

Ultrasonic sensors are used effectively to measure the rate of flow in many biomedical and industrial researches. The main structural element of the ultrasonic sensor is a piezoelectric emitter of the short acoustic waves. The frequencies lying outside the audible acoustic range – in an ultrasonic field are used to measure the flow rate. The principle of work of the ultrasonic flow sensors is based on one of the two physical principles. The sensors of the first type (measurement of the signal times) are based on the velocity addition law: the speed of sound propagating in a moving medium is equal to the sum of the speed of sound relative to the medium and the speed of the medium itself. The sensors of the second type are based on the change of frequency (Doppler shift) of ultrasonic waves scattered by the particles in a moving medium.

The electroacoustic transducers from the piezoelectric materials, carrying out the conversion of electrical power into acoustic oscillations, are used in ultrasonic flow meters. The ideal piezoelectric material for electroacoustic transducer is a material that provides low noise, high efficiency of conversion and allows creating the converter with high quality factor.

Selection of the operating frequency of the converter is determined by fundamental physical factors. The final value of the transducer diameter gives rise to a diffraction distribution of the ultrasonic wave intensity, similar to the aperture diffraction in optics. The boundaries of the emitted beam for different converter diameters and frequencies are shown in Fig. 7.1.9.

![Fig. 7.1.9. The near-and far-field regions for different converter diameters and frequencies](image)

In the near-field region the beam has a substantially cylindrical shape corresponding to the geometry of the emitter and its broadening is small. However, the intensity distribution in the beam is uniform, because here there are numerous interference maxima and minima. In the far-field region the beam diverges, the intensity of the sound wave in the beam is inversely proportional to the square of the distance from the transmitter.
The effect of the beam divergence affects the spatial resolution so the far-field region is not recommended to use, big converters and high operating frequencies are needed for operation in the near-field region. In industrial applications the high spatial resolution of flow rate measurements can be obtained by selecting the operating frequency and the size of converter, so that the size of near-field region will be approximately adequate to flow wires (pipes, pipelines etc.) diameter.

The correct choice of operating frequency is very important for the measurement of blood flow. For a beam with a constant cross-section power of ultrasonic wave decreases exponentially with distance due to its absorption into the tissue. From this viewpoint low operating frequencies are preferred, since the absorption coefficient of ultrasound increases in quasi-linear manner with frequency increase. On the other hand, the most common ultrasonic flow meters – Doppler flow sensors – are working on the principle of detecting the power of ultrasonic wave scattered by moving red blood cells, and the power dissipation is proportional to the fourth power of the frequency. Thus, in these flow meters in order to increase a detectable power, the operating frequency should be increased. The compromise is achieved at the operating frequency in the range of 2 to 10 MHz. Flow sensors working on the principle of measuring the sound running time, are widely used in industry and are also suitable for the measurement of blood flow.

Two possible ways of the transducers location in the sensor of this type are illustrated in Fig. 7.1.10. Method for arranging shown in Fig. 7.1.10a has the obvious advantage of being able to fix the converters on the outer surface of the tube or blood vessel, which prevents flow restriction. Transducers isolated from the tube, which are used for high temperature measurements (for example, coal gasification), are shown in Fig. 7.1.10b.

![Fig. 7.1.10 Options for the transducers location in the flow meters working on the principle of measuring the sound running time](image-url)
In this case, the relationship of converters with media is carried out by means of waveguides. For the flow meter configuration shown in Fig. 7.1.10b, effective ultrasonic velocity in the blood vessel or tube equals the sum of sound velocity relative to the fluid and components related to the magnitude of \( u \) – flow rate averaged along the ultrasonic wave propagation path. The formula that determines the time of the ultrasonic signal propagation between the transducers is given by:

\[
t = \frac{D}{c \pm u \cos \theta},
\]

where \( D \) – the distance between the transducers; \( c \) – the velocity of ultrasonic waves propagation; \( u \) – the flow rate.

The version of ultrasonic flow sensor based on the principle of measuring the time of the signal, that is used in some industrial applications is shown in Fig. 7.1.10b. The speed of sound can vary with temperature, and this may involve a significant error in the signal propagation time measurement \( t \).

The most standard flow sensor working on the principle of measuring the sound propagation time, performed according to the scheme shown in Fig. 7.1.10a. The advantages of such flow sensors are the following:

- They can measure the flows of various liquids and gases, because the measurement is not needed the presence particles reflecting the ultrasound in the fluid;
- They allow to define the direction of the flow;
- Their indications are relatively insensitive to changes in viscosity, temperature and density of the fluid;
- From all of the commercially available flow meters, industrial devices of this type are of highest accuracy.

The considered flow sensors are suitable for measuring the flow of liquids in many industrial plants. The group of fluids these sensors can operate includes milk, butter, treated wastewater, pharmaceutical liquids, and liquid paper pulp.

In the Doppler flow meters of continuous action the effect of changing the frequency of the sound detected by a moving receiver, receding from a stationary source, is used (Doppler effect).

Fig. 7.1.11 shows how the transducers can be located in the Doppler flow sensors of continuous action.
If the emitter and the receiver are stationary and the object reflecting the ultrasonic wave (the particle in a fluid medium) is moving (Fig. 7.1.11), then caused by the Doppler effect frequency shift in a symmetrical arrangement of transducers with respect to the axial-symmetric flow is calculated by the formula:

$$v_D = \frac{2v_0u \cos \theta}{c}$$  \hspace{1cm} (7.1.16)

where $v_D$ – Doppler frequency shift; $v_0$ – radiated ultrasonic wave frequency; $u$ – velocity of an object (particle in a fluid medium); $c$ – sound velocity; $\theta$ – angle between the direction of ultrasonic wave radiation and the axis of the pipe or blood vessel. If the flow does not have axial symmetry or transducers are located asymmetrically (as in Fig. 7.1.11b), then the formula (7.1.16) needs to introduce an additional trigonometric ratio.

![Diagram of transducers in Doppler flow sensors](image)

*Fig. 7.1.11. Location of transducers in Doppler flow sensors of continuous action: a – the transmitter and the receiver are on opposite sides of the wire thread, b – both of the transmitters are on the one hand of the wire thread*

When using a Doppler flow sensor of continuous action to obtain a Doppler shift signal, the presence of any particles in fluid is needed. Doppler shift signal is not a single frequency harmonic signal, which is due to several reasons:

- Velocity distribution profile over the cross section of flow (the flow profile) is heterogeneous. The particles move at different speeds, generating different Doppler shifts in frequency.
- The particle reflects the ultrasonic wave in a short period of time.
- The chaotic rotation of the particles and turbulence cause different Doppler shifts.
Two other lacks of Doppler flow sensors of continuous action are: almost complete absence of information about the flow profile and the inability to determine the direction of the flow rate without any additional signal processing.

Method of laser anemometry: the frequency of the laser radiation scattered on the smallest particles moving with the flow is proportional to velocity of the particles, is also based on Doppler effect.

Particles-radiation diffusers are almost always present in a fluid of liquid or gas (aerosols, insoluble impurities or gas bubbles in the liquid). If it is necessary the polystyrene particles of about 0.1 µm may be added in the flux.

The frequency is measured in the experiment and from known values of the laser radiation wave length $\lambda$ and the device geometric parameter the local flow velocity is determined.

Compared with other methods laser anemometry has some advantages: absence of disturbances typically introduced by the sensor into the flow, a high spatial resolution ($10^{-3}$ mm$^3$), linear relationship between the Doppler signal with a rate of flow, lack of need for calibration, the sensitivity to the direction of the flow. It should, however, bear in mind that in measurements in non-isothermal flows the distortions of laser beams due to inhomogeneity of the field of the material density are occurred.

Pulse Doppler flow sensor provides information about the fluid flow profile. The principle of this device operation is shown in Fig. 7.1.12. Transducer connected to the generator emits a signal of a certain frequency. This transducer performs the function of the emitter and the receiver, reflected signal with a Doppler shift is taken with a certain time delay after the moment of the primary signal radiation.

Fig. 7.1.12. The principle of pulsed Doppler flow meter operation
The time interval between the moments of emission and reception of the signal is a direct pointer of the distance to the reflecting particles. Consequently, full «sweep» of the signal reflections across tube or blood vessel can be obtained. The velocity profile in the cross section of blood vessel is obtained by registration of the signal Doppler shift at different time delays. With a pulsed Doppler flow sensor blood vessel diameter can be estimated. As is evident from Fig. 7.1.12, received signals A and C are due to reflections from the near and far vessel walls respectively. The distance between points where these reflections take place is directly related to the vessel diameter.

A similar principle of measurement is in the basis of ultrasonic scanning method in amplitude mode and method of echo cardiography. The ultrasound transducer is installed in front part of the body or organ to be scanned. This transducer emits an ultrasonic signal experiencing reflection on any tissue inhomogeneity along the scanning direction. The delay between emission time and signal reception time can be used to determine the localization of this heterogeneity along a certain scanning path. The duration of the emitted pulse is an important factor in the use of pulsed Doppler sensor for registration the blood flow. Ideally, this should be a very short pulse to get a good resolution of the distance. On the other hand, to achieve a sufficiently high value of signal / noise ratio and good resolution for velocity, the duration of the pulse must be large enough. Doppler measuring systems operating in a pulsed mode are inherent in the internal limit. It is expressed in the fact that for a given range to the range of measured velocities is limited. This forces the use of pulses with less repetition rate.

This means that it is impossible to measure high rates over long distances to the reflecting object.

The main advantage of pulsed Doppler flow sensors is the possibility of obtaining information about the flow profile.

7.2. The viscosity of liquids measurement

The study of hydrodynamic and thermal interaction between different bodies moving with the flow of liquids and gases is of great practical importance in many fields of science, technology, production: energetics, aviation, aerospace, transportation of liquids and gases, etc. The theoretical basis of this research is a system of differential transport equations: the equations of motion, energy, and continuity.
One of the most important characteristics in these equations is the viscosity of the medium – the property of real liquids and gases to resist the tangential forces efforts. If the shear force attached to any surface of a limited volume of fluid (gas), the frictional forces, which action will cause the velocity gradient and its peak will occur at the point of application of force, arise between the layers of the medium. Newton equation for viscous fluids in the one-dimensional problem has the form:

$$\tau = \frac{dF}{dS} = -\eta \frac{dV}{dy},$$  \hspace{1cm} (7.2.1)

where \(dF\) – viscous force acting on the area \(dS\) of the layer surface; \(\eta\) – dynamic viscosity (internal friction coefficient) of liquid; \(\tau\) – shear stress.

For two-dimensional flow

$$\tau_{xy} = \eta \left( \frac{\partial v_x}{\partial y} + \frac{\partial v_y}{\partial x} \right).$$  \hspace{1cm} (7.2.2)

Thus, viscosity is a measure of the fluid (or gas) internal friction forces and is numerically equal to the friction force between the two layers with an area equal to one at a velocity gradient equal to unity.

In the CGS system the unit of viscosity measurement is the value of \([\eta] = \text{dyne}\cdot\text{s/cm}^2 = \text{g/cm}\cdot\text{s},\) called poise (P). For low-viscosity liquids used hundredth of a poise – centipoise (cP).

The SI unit of viscosity has no special name. As follows from (7.2.1), \([\eta] = \text{N}\cdot\text{s/m}^2 = \text{Pa}\cdot\text{s}.

In hydrodynamics the kinematic viscosity, defined as the ratio of the dynamic viscosity to the density of the medium, is widely used:

$$\nu = \frac{\eta}{\rho}.$$  \hspace{1cm} (7.2.3)

The unit of kinematic viscosity in the CGS system is the value of \([\nu] = \text{cm}^2/\text{s},\) called Stokes (St). In the SI system \([\nu] = \text{m}^2/\text{s}.

Dynamic viscosity of gases, which is derived from the molecular-kinetic theory, is:
\[ \eta = \frac{mn\langle v \rangle l}{3} = \frac{1}{3} \rho \langle v \rangle l, \]  

where \( \langle v \rangle \) – average velocity of the molecules; \( l \) – average length of the molecule free path; \( n \) – number of molecules per unit volume.

The dependence of the gas viscosity on temperature is quite significant: the temperature increases, the kinematic viscosity increased. The viscosity of gases is strongly dependent on the pressure only in some areas of pressure and temperature. At moderate pressures and temperatures, the effect of pressure on the viscosity can be ignored. Near the line of vapour saturation and the critical point, pressure has a significant effect on the viscosity: the pressure increases, the viscosity increased.

Viscosity of gases can be calculated by using methods based on theoretical assumptions, but for determining the viscosity of liquids the similar theoretical framework doesn’t exist. According to their properties liquids are similar both with gases and with solids. Like gases, liquids take the form of container in which they are located. Like liquids, they have a relatively high density and they are difficult to compress. The dual nature of the fluids properties is associated with the peculiarities of their molecules motion. In liquids, as well as in crystals, each molecule is in a potential well of the electric field generated by the surrounding molecules. Molecules oscillate with the average frequency close to the oscillation frequency of the atoms in the crystalline solids, and with amplitude determined by the dimensions of the volume provided by adjacent molecules. The potential well depth greater than the average kinetic energy of oscillating molecule, so the molecules fluctuate around a more or less stable equilibrium positions. However, the difference between these two energies is small in liquids, so that the molecules often leave «their» potential well and take the place of the other. Unlike solids, there are free places in liquids – holes, so the molecules can move, leaving one position and taking up one of the adjacent holes. Thus, the molecules move slowly inside the liquid, staying part of the time near certain equilibrium positions and form the picture of the space lattice changing in time. This character of the molecules motion explains both the low rate of diffusion in liquids and a bigerrviscosity (compared to gas). In gases the viscosity is explained by the directed transfer of momentum by molecules in their thermal motion. In liquids such transitions are significantly slowed.

The values viscosity coefficients of liquids are significantly different from those of gases – they are a lot larger in magnitude and decrease sharply with increasing temperature.
In a wide temperature range from the region slightly above the normal boiling point to the region close to freezing point of liquid viscosity changes according to the law:

$$\eta \sim A \cdot e^{B/T},$$  

(7.2.5)

rapidly decreasing with temperature increase.

However, at low temperatures, and also at reduced temperature values above 0.7 – 0.75, this dependency is not satisfied.

Liquids viscosity at temperatures below the normal boiling point usually doesn’t depend on medium pressure, but at high pressures there is a significant increase in viscosity with pressure increase.

Due to the theory of fluid viscosity weak development the basic information about these important properties of liquids are obtained by the experiment.

Liquids, for which the dependence (7.2.1) is valid, are called normal or Newtonian.

The dynamic viscosity of these fluids depends on temperature and pressure and doesn’t depend on shear rate. A plot of shear stress on the velocity gradient $d\omega /dy$ (flow curve) is a straight line, which slope is numerically equal to the dynamic viscosity $\eta$.

Newtonian behaviour is inherent for liquids, in which the viscous dissipation of energy caused by the collision of small molecules. All gases, liquids and solutions of low molecular weight form Newtonian fluid group.

The liquid viscosity is determined by means of an instrument called viscometers. There are several types of viscometers, which differ in their construction and principle of operation. The main ones are the capillary viscometer, the expiration viscometer and the rotational viscometer.

### 7.2.1. Capillary viscosimeters

In capillary viscometers viscosity of the liquid is determined by observation of the sample liquid movement through the tube of very small diameter, which establishes a laminar flow regime. Mentally distinguish the cylinder of radius $r$ and length $L$ in a liquid (Fig. 7.2.1):
The fluid pressure at the ends of cylinders is equal to $p_1$ and $p_2$. At steady-state flow pressure force on the cylinder $(p_1 - p_2)\pi r^2$ is balanced by the friction force $F_{fr} = S\eta \frac{\text{d}u}{\text{d}r}$, acting on the cylinder from the outer fluid layers, so

$$(p_1 - p_2)\pi r^2 + S\eta \frac{\text{d}u}{\text{d}r} = 0,$$  \hspace{1cm} (7.2.6)

or, since $S = 2\pi rl$,

$$(p_1 - p_2)\pi r^2 + 2\pi rl\eta \frac{\text{d}u}{\text{d}r} = 0.$$  \hspace{1cm} (7.2.7)

Integrating (7.2.7), we obtain

$$u = -\frac{p_1 - p_2}{4\eta l} r^2 + c.$$  \hspace{1cm} (7.2.8)

The constant of integration can be found from the boundary conditions: at the tube radius $R$, where the liquid «sticks» to the walls, the fluid velocity equals 0. From this condition:

$$c = \frac{p_1 - p_2}{4\mu l} R^2.$$  \hspace{1cm} (7.2.9)

Substituting the value of the constant in (7.2.8), we obtain:

$$u = \frac{p_1 - p_2}{4\eta l} (R^2 - r^2) = \frac{p_1 - p_2}{4\eta l} R^2 \left(1 - \frac{r^2}{R^2}\right).$$  \hspace{1cm} (7.2.10)

Speed value on the axis of pipe equal to:
\[ \nu_0 = \nu(0) = \frac{p_1 - p_2}{4\eta l} R^2. \]  

(7.2.11)

Taking this into account, the formula (7.2.10) can be written in the form:

\[ \nu(r) = \nu_0 \left(1 - \frac{r^2}{R^2}\right). \]  

(7.2.12)

Thus, at laminar current velocity varies with distance from the tube axis parabolically (see Fig. 7.2.2):

![Fig. 7.2.2. At laminar current velocity varies with distance from the tube axis parabolically](image)

Assuming the current is laminar; let us compute the fluid flow \( J \), i.e. the volume of fluid flowing through the pipe cross-section in unit time. We partition cross-section of the pipe on the ring width \( dr \) (Fig. 7.2.3). The volume of fluid equal to the product of the area of the ring \( 2\pi rd r \) on the flow rate at points separated by a distance \( r \) from the axis of the tube will pass through the ring of radius \( r \) per second. Taking into account the formula (7.2.12), we get:

\[ dJ = \nu_0 \left(1 - \frac{r^2}{R^2}\right) 2\pi r dr. \]  

(7.2.13)

To get the full flow \( J \), we must integrate the expression (7.2.13) over the variable \( r \) in the range of zero to \( R \):

\[ J = \int_0^R \nu_0 \left(1 - \frac{r^2}{R^2}\right) 2\pi r dr = \frac{1}{2} \pi R^2 \nu_0 = \frac{1}{2} Sv_0, \]  

(7.2.14)

where \( S \)– the area of the pipe.
Substituting into (7.2.14) the value (7.2.11), we obtain a formula for the flow:

$$J = \frac{(p_1 - p_2)\pi R^4}{8\eta l}.$$  \hspace{1cm} (7.2.15)

This equation is called the Poiseuille formula. According to (7.2.15), the flow of fluid is proportional to the pressure drop per unit length of the pipe, the fourth power of the radius of the pipe and is inversely proportional to the coefficient of viscosity. Poiseuille formula is valid for laminar fluid flow in the absence of slip at the liquid – capillary wall.

Measuring the flow rate, pressure drop, the length of the tube and its radius, the fluid viscosity can be determined from the Poiseuille equation.

In capillary viscometers, fluid from one vessel flows across the capillary of the determined cross-section and length to another vessel under the influence of a pressure difference. These vessels have many times larger cross-section than the capillary, so the velocity of the fluid in the vessels is ten times less than in the capillary, and not all the pressure will go to overcome the viscous fluid friction. Part of it will be spent on a transfer of a certain amount of the kinetic energy to the fluid. Therefore, an amendment for the kinetic energy is moved in the Poiseuille equation (Hagenbach correction).

In the investigation of organic liquids with a kinematic viscosity greater $10^{-6}$ m²/s the amendment is of a percent. And because of the low kinematic viscosity of liquid metals ($\sim 10^{-7}$ m²/s) amendment could reach 10 – 15 %. An amendment for the kinetic energy can be established experimentally by taking measurements at different values of the pressure difference $p$. During this process the time at which the fluid volume $V$ flows throw the capillary with radius $R$ is determined.

The second amendment allows for the «end» effect, it reflects the influence of the capillary initial section on the fluid motion nature. At the interface of the capillary with reservoir, from which the fluid flows, there may be a screw motion and twist. This effect is
taken into account by introducing the apparent length $l'$ instead of the true length of the capillary $l$: $l' = l + nR$, where $n$ – coefficient approximately equal to one, it is determined experimentally on the basis of measurements at different values of $l$.

This method has been widely spread in viscometry of liquids at room and high temperatures, due to the simplicity and the possibility of obtaining the absolute values of viscosity. If the temperature of the melt is high, there is a difficulty in finding the material of the viscometer (it doesn’t have to interact with the environment under the study and change its own shape). Since the calculation formula includes $R^4$, then the smallest diameter changes or inaccuracy of its measurement gives greater error in the determination of the viscosity. Typically quartz capillaries, sometimes platinum are used. Accuracy of the liquid metal viscosity measurements by capillary expiration method depends on the capillary radius and length, of the flow time, pressure and volume of the vessels measuring accuracy and amounts to $0.5 - 2.0\%$ depending on the temperature range. The limits of measuring the viscosity are $0.01 - 0.1\ Pa\cdot s$.

In practice, the determination of the viscosity is usually done by comparing the cost or time of equal volumes of the two fluids expiration: the investigated and a standard, whose viscosity is known, in two identical capillary tubes, all other things being equal. This principle is based on the structure of the Zhukovsky capillary viscometer, the calculating formula for which is:

$$\eta_i = \eta_2 \frac{V_1}{V_2},$$

(7.2.16)

where $\eta_i$ – the liquid under investigation viscosity; $\eta_2$ – the viscosity of the standard (reference) liquid; $V_1$ and $V_2$ – volumes of the fluids flowing through the capillary tube of the same length and diameter for the same time.

### 7.2.2. Rotational viscosimeters

The essence of methods is in the fact that test liquid is placed in a gap between two surfaces of a regular geometric shape. One of the surfaces is driven to rotate at a constant speed. In this case rotational movement is transmitted to the other surface by the liquid. According to the method theory, the absence of liquid slip near surfaces is assumed. The torque transmitted from one surface to another, is a measure of viscosity. To measure the torque, second surface is connected to the dynamometric device. A small gap between the
surfaces is required to shift the deformable fluid. Viscometers of this group can produce measurements of liquids with viscosities up to $10^3$ Pa·s.

If the inner cylinder is suspended by an elastic thread, the upper end of which is fixedly secured, the thread is twisted at an angle $\varphi$. In this case torque due to viscosity of liquid is balanced by moment of filament elastic forces.

Now electric rotary viscometers are widely used: the rotation of the inner cylinder immersed in a fluid, carried out by an AC or DC power. When cylinder rotating at a constant speed submerged in a medium, the braking torque that occurs at the motor shaft is directly proportional to the viscosity of the fluid. The changing of the braking torque causes a corresponding change in the electrical characteristics of the motor, which are usually detected by a bridge circuit. Electric rotational viscometers operate within 0.01–10 Pa·s with accuracy ± 2 – 5 % depending on the viscosity values limits and temperature conditions.

In a rotational viscometer «Reotest» mechanism of measuring instrument is a torque mechanical-electrical transducer acting on the inner tube of the coaxial cylinders (Fig. 7.2.4). By measuring shaft the internal cylinder is connected to the helical spring, which deviation is a measure of the torque $M$.

Fig. 7.2.4 The rotational viscometer schematic diagram, where 1 – fixed cylinder, 2 – rotating cylinder, 3 – thermostat.

In rotational viscometers test liquid is placed in a gap between two long vertically spaced coaxial cylinders. One of them is driven to rotate at variable angular speed, while the other cylinder undergoes a twisting force, which magnitude is measured during the experiment. Changing of torque depending on the number of revolutions of the rotating cylinder can be interpreted as the relationship between shear stress and shear rate.
Changing of shear rate at each point of the sample depends on the width of the annular gap between the cylinders. If the gap is sufficiently small, the change in shear rate across the gap is insignificant, i.e. radial variation of this magnitude would be negligible. Liquid viscosity value in these viscometers defined by the rotational speed of the movable cylinder at a predetermined torque or vice versa, a torque calling predetermined speed:

\[ \eta = k \frac{M}{\omega}, \]  

(7.2.17)

where \( M \) – torque; \( \omega \) – angular velocity; \( k \) – device constant.

### 7.2.3. The falling ball method

In practice of hydrodynamic investigations viscometers, in which the viscosity is determined by the time of the uniform ball falling in a transparent vertical tube filled with the test liquid, are also widely used. The method theory is based on Stokes law. By its viscosity, the body moving in a fluid entrains surrounding liquid layers and therefore experiences resistance (friction) from the liquid. The resistance force depends on the speed of the body, its size and shape. As was established by English physicist and mathematician Stokes, for bodies of spherical shape moving at a low speed, the liquid resistance force \( F \) is proportional to the viscosity of the fluid \( \eta \), the radius of the sphere \( r \) and speed \( \nu \):

\[ F = 6\pi \eta \nu . \]  

(7.2.18)

Formula (7.2.18), called the Stokes law, is also applicable to movement of spherical bodies in a gas, for example, to the case of rain drops falling in the atmosphere.

Three forces are acting on the ball of mass \( m \) and radius \( r \) falling at a rate \( \nu \) in liquid with viscosity \( \eta \): gravity \( F_1 \), buoyancy force of liquid \( F_2 \) and liquid resistance force \( F \) (Fig. 7.2.5).

![Fig. 7.2.5 The forces acting on the ball falling in the liquid](image)
Since the forces $F_1$ and $F_2$ are constant and the force $F$ increases with the speed of the ball, then at some point in time, these forces will cancel each other:

$$F_1 = F_2 + F.$$  \hfill (7.2.19)

Then the ball will move uniformly.

$$F_1 = mg = \frac{4}{3} \pi r^3 \rho_1 g ,$$ \hfill (7.2.20)

$$F_2 = \frac{4}{3} \pi r^3 \rho_2 g ,$$ \hfill (7.2.21)

where $\rho_1$ – density of the sphere; $\rho_1$ – density of the liquid and $g$ – gravity acceleration.

Therefore,

$$\frac{4}{3} \pi r^3 \rho_1 g = 6\pi \eta \nu + \frac{4}{3} \pi r^3 \rho_2 g ,$$ \hfill (7.2.22)

whence after appropriate transformations we obtain an expression by which the liquid viscosity can be empirically determined:

$$\eta = \frac{2r^2 g (\rho_1 - \rho_2)}{9\nu} .$$ \hfill (7.2.23)

This equation is valid for small velocities of the bodies. For the finite size of the cylindrical vessel corrections are applied. This method is used mainly for viscosity measurements of molten glass and salts within the viscosity from 1.0 to $10^4$ Pa·s. When using the method the difficulty is in determining the ball location in the opaque melt, as well as in selection of materials for the balls manufacture. Usually, to simplify the calculations a device calibration is produced for fluids with known viscosities.
8. Introduction to the heat conduction theory

Nina Karbalevich
Belarusian State University, Physical Faculty
**Introduction**

Heat conduction – the process of the heat transfer that can be carried by conduction, convection and radiation. If the heat is transferred by all three methods, we have a complex heat transfer. In many cases the only one transport mechanism is dominant, such as thermal conductivity.

The process of heat conduction – is the transfer of heat by the atomic and molecular level, atoms, molecules, electrons and phonons in the presence of temperature gradients. The mechanism of the transfer depends on the physical state of matter.

The basic heat conduction law is Fourie law:

\[ \dot{q} = -\lambda \text{grad}T, \]  

(8.1)

that is, the heat flux density is directly proportional to the temperature gradient.

The proportionality factor \( \lambda \) called the coefficient of thermal conductivity. It is numerically equal to the amount of heat passing through the pad unit area per unit time in the direction normal to that site at a single temperature gradient.

Heat transfer processes occurring in space and time. Therefore, the temperature is a function of position and time: \( T = f(x, y, z, \tau) \).

The set of instantaneous temperature at all points in space studied by is called the temperature field. The temperature depends in general on the three coordinates and time. If the temperature depends on the time, the field is called a non-stationary. If the temperature field is not changed in time, i.e. is a function of the coordinates \( T = f(x, y, z, \tau) \), it is called steady.

The temperature may depend on three, two, one coordinate. In accordance with this, three-dimensional, two-dimensional and one-dimensional temperature fields are distinguished.

One of the problems of the theory of thermal conductivity, energy physics, building thermal physics and other sciences is the calculation of temperature fields and heat flux in the bodies of various shapes under different conditions of interaction of these objects with the environment. To do this, the corresponding differential equations have to be solved.
8.1. The differential equation of heat conduction

8.1.1. Boundary conditions

In the derivation of the heat conduction differential equation we use the following assumptions:

- The body studied is homogeneous and isotropic;
- Its physical parameters are constant;
- The considerate body deformation associated with the temperature change is very small compared with the volume of the sample;
- Internal sources are evenly distributed in the body.

The derivation of the heat differential equation is based on the law of energy conservation: the amount of heat supplied to the external volume element per unit time by heat conduction and the action of internal heat sources, is equal to its internal energy or enthalpy (depending on the process under consideration: an isobaric or isochoric) change in unit time.

\[ dQ_1 + dQ_2 = dQ, \quad (8.1.1) \]

where \( dQ_1 \) – the flow of heat introduced into the elementary volume; \( dQ_2 \) – the flow of heat generated in unit volume as a result of internal heat sources; \( dQ \) – change in the internal energy (or enthalpy) of the elementary volume.

We distinguish in this sample the elementary volume (Fig. 8.1.1), the sides of which are \( dx, dy, dz \).

![Fig. 8.1.1 The elementary volume](image)

The heat passing through the left edge in the unit time is:
\[ dQ_x = q_x dydz. \]  

(8.1.2)

The amount of heat escaping through the opposite face of a unit time, equal to

\[ dQ_{x+dx} = q_{x+dx} dydz. \]  

(8.1.3)

If \( q_x > q_{x+dx} \), then the unit volume will be heated.

We expand the \( q_{x+dx} \) in a Taylor series:

\[ q_{x+dx} = q_x + \frac{\partial q_x}{\partial x} dx + \frac{\partial^2 q_x}{\partial x^2} \frac{(dx)^2}{2!} + \ldots. \]  

(8.1.4)

If we restrict ourselves by the first two members of the series, then:

\[ q_{x+dx} = q_x + \frac{\partial q_x}{\partial x} dx. \]  

(8.1.5)

The amount of heat supplied to the considered volume by conduction per unit time in the direction of the axis OX:

\[ dQ_x - dQ_{x+dx} = -\frac{\partial q_x}{\partial x} dx dydz. \]  

(8.1.6)

The amount of heat supplied to the elementary volume per unit time in all directions, equal to:

\[ dQ_1 = -\left( \frac{dq_x}{\partial x} + \frac{dq_y}{\partial y} + \frac{dq_z}{\partial z} \right) dx dydz. \]  

(8.1.7)

The amount of heat generated by the internal springs in the elementary volume per unit time, equal to:

\[ dQ_2 = w_v dx dydz, \]  

(8.1.8)

where \( w_v \) – power of internal heat sources.

During the isochoric process the heat supplied to the elementary volume goes to internal energy of the body changing:
Using the expressions (8.1.7 – 8.1.9), we obtain:

\[ c_v \rho \frac{\partial T}{\partial \tau} = - \left( \frac{\partial q_x}{\partial x} + \frac{\partial q_y}{\partial y} + \frac{\partial q_z}{\partial z} \right) + w_v, \quad (8.1.10) \]

or

\[ c_v \rho \frac{\partial T}{\partial \tau} = - \text{div} \bar{q} + w_v. \quad (8.1.11) \]

Equations (8.1.10) and (8.1.11) called the energy equation for the isochoric process.

For isobaric process

\[ dQ_1 + dQ_2 = dH \]

\[ dH = c_p \rho \frac{\partial T}{\partial \tau} \, dx \, dy \, dz = \rho \frac{\partial h}{\partial \tau} \, dx \, dy \, dz', \quad (8.1.12) \]

where \( H \) – enthalpy of the system (\( H = U + pV \));\( h \) – specific enthalpy. The differential equation of energy for isobaric process can be represented as:

\[ \rho \frac{\partial h}{\partial \tau} = - \left( \frac{\partial q_x}{\partial x} + \frac{\partial q_y}{\partial y} + \frac{\partial q_z}{\partial z} \right) + w_v \quad (8.1.13) \]

or

\[ \rho \frac{\partial h}{\partial \tau} = - \text{div} \bar{q} + w_v. \quad (8.1.14) \]

In solids, heat transfer is performed by Fourier law:

\[ q_x = -\lambda \frac{\partial T}{\partial x}, \quad q_y = -\lambda \frac{\partial T}{\partial y}, \quad q_z = -\lambda \frac{\partial T}{\partial z}. \quad (8.1.15) \]

The value of the difference of the specific heats \( c_p \) and \( c_v \) of solids is small, and we can take \( c_p = c_v = c \). So from (8.1.10) we obtain:
\[
\frac{\partial T}{\partial \tau} = \frac{1}{cp} \left( \frac{\partial}{\partial x} \left( \lambda \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial y} \left( \lambda \frac{\partial T}{\partial y} \right) + \frac{\partial}{\partial z} \left( \lambda \frac{\partial T}{\partial z} \right) \right) + \frac{w_v}{cp} \quad (8.1.16)
\]

or

\[
\frac{\partial T}{\partial \tau} = \frac{1}{cp} \text{ div } \lambda \text{ grad } T + \frac{w_v}{cp} . \quad (8.1.17)
\]

Equations (8.1.16) and (8.1.17) are called differential equations of heat conduction. They establish a connection between the temporal and spatial variations of temperature at any point in the body in which the process of heat conduction occurs.

If the thermal characteristics of the sample under consideration are constant, then the differential heat conduction equation is transformed into:

\[
\frac{\partial T}{\partial \tau} = \frac{\lambda}{cp} \left( \frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} \right) + \frac{w_v}{cp} . \quad (8.1.18)
\]

Since the connection of heat and thermal conductivity coefficients is represented as:

\[
\frac{\lambda}{cp} = a , \quad (8.1.19)
\]

where \(a\) – thermal conductivity coefficient and \(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} = \nabla^2 T\), then we get

\[
\frac{\partial T}{\partial \tau} = a \nabla^2 T + \frac{w_v}{cp} . \quad (8.1.20)
\]

If the body has no internal heat sources, i.e. \(w = 0\), then

\[
\frac{\partial T}{\partial \tau} = a \nabla^2 T . \quad (8.1.21)
\]

If the body has internal source of heat, but heat conduction is stationary, the differential equation of heat conduction becomes a Poisson equation:

\[
\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} + \frac{w_v}{\lambda} = 0 . \quad (8.1.22)
\]
At steady-state heat conduction process and the absence of internal heat sources, equation (8.1.20) has the form of the Laplace equation:

\[
\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} = 0. \tag{8.1.23}
\]

Thermal conductivity coefficient is essential for non-stationary processes and characterizes the rate of temperature change. It is a measure of thermal inertial properties of bodies. The temperature change with time at any point in space is proportional to the coefficient of thermal conductivity. Therefore, temperature equalization at all points of the sample will occur quickly in the body, which has a high coefficient \(a\). Coefficient \(a\) depends on the nature of the substance. For example, liquids and gases have a high thermal inertia and, consequently, a small \(a\). In metals, there are low thermal inertia and high thermal conductivity.

We define the physical meaning of the thermal conductivity. We denote the internal energy of the body per unit volume \(u_v\).

\[
\frac{\partial T}{\partial n} = \frac{\partial T}{\partial u_v} \frac{\partial u_v}{\partial n} = \frac{1}{c_v \rho} \frac{\partial u_v}{\partial n}. \tag{8.1.24}
\]

The amount of heat flux can be represented as:

\[
q = -n \lambda \frac{\partial T}{\partial n} = -n \frac{\lambda}{c_v \rho} \frac{\partial u_v}{\partial n} = -a_v \nabla u_v. \tag{8.1.25}
\]

At steady-state one-dimensional heat flow

\[
\frac{Q}{St} = a_v \frac{u_{v_2} - u_{v_1}}{x_2 - x_1}. \tag{8.1.26}
\]

Thus, the coefficient \(a\) is numerically equal to the quantity of heat that passes in unit time through unit area of the unit surface with a gradient of the volume density of the internal energy. \([a_v] = \frac{J \times m^4}{m^2 \times s \times s} = \frac{m^2}{s}.\]

At constant pressure thermal conductivity coefficient characterizes the enthalpy of the body:

\[
a_p = \frac{\lambda}{c_p \rho}; \quad C_p = c_p \rho = \frac{\partial h}{\partial T} \quad \rho; \quad q = -n \lambda \frac{\partial T}{\partial h} \frac{\partial h}{\partial n} = -a_v \nabla h \tag{8.1.27}
\]
Thermal conductivity coefficient is the diffusion coefficient of internal energy or enthalpy depending on the coupling conditions of the body and the environment.

We define the physical meaning of the thermal conductivity coefficient, using the equation of isothermal surface: \( T(x, y, z, \tau) = \text{const} \). The total differential of this function is:

\[
\frac{\partial T}{\partial \tau} d\tau + \frac{\partial T}{\partial n} dn = 0 \quad \text{or} \quad \frac{\partial T}{\partial \tau} + \frac{\partial T}{\partial n} \frac{dn}{d\tau} = 0,
\]

where \( \nu = \frac{dn}{d\tau} \) – speed of movement or velocity of propagation of an isothermal surface.

From the differential equation of heat conduction and the last equation it follows that:

\[
a = -\nu_T \frac{\nu_T}{\nu^2 T}.
\]

We denote the \( \frac{NT}{\nu^2 T} = A \). Then:

\[
a = \nu_T A.
\]

Consequently, thermal conductivity coefficient is proportional to the velocity of propagation of an isothermal surface.

Coefficient \( a \) depends on temperature, and for porous bodies – on density and humidity. Therefore, only as an approximation coefficients \( a \) and \( \lambda \) can be considered constant.

The differential equation describes a class of phenomena of heat conduction. Particular features that are describing the specific heat conduction together with the differential equations are called the terms of the uniqueness or boundary conditions.

Terms of uniqueness include:

- Geometric conditions that characterize the shape and size of the body;
- Physical conditions that characterize the physical properties of the environment and the body;
- Temporary (initial) conditions characterizing the temperature distribution in the studied body at the initial time;
- Boundary conditions that characterize the interaction of the body in the environment.
The shape and linear dimensions of the body in which the process takes place are specified by geometric conditions.

The physical parameters of the body ($\lambda$, $c$, $\rho$, etc.) are specified by physical conditions.

The initial conditions are in setting the law of the temperature distribution within the body at the initial time:

$$T(x, y, z, 0) = f(x, y, z). \quad (8.1.31)$$

With a uniform temperature distribution in the body at the initial time:

$$T(x, y, z, 0) = \text{const} . \quad (8.1.32)$$

The boundary conditions can be defined in various ways.

1) **The boundary conditions of the 1st kind** are setting the temperature distribution on the surface of the body at any time:

$$T_n(\tau) = f(\tau). \quad (8.1.33)$$

In the particular case $T_n(\tau) = T_c = \text{const}$, i.e. the surface temperature is constant throughout the process of heat transfer. This can be accomplished by artificially maintaining a constant temperature or under special conditions of heat transfer between the environment and the body surface.

2) **The boundary conditions of the 2nd kind** are setting the heat flow density for each point on the surface of the body as a function of time:

$$q_n(\tau) = f(\tau). \quad (8.1.34)$$

In the particular case of constant heat flux: $q_n(\tau) = q_c = \text{const}$. Such a case occurs when heat heating metal products in high temperature furnaces, where heat transfer takes place mainly by radiation to the Stefan-Boltzmann law.

3) **The boundary conditions of 3rd kind** characterize the heat exchange between the surface of the body and the environment. For a description of the process Newton-Richman law is used:

$$q_s = \alpha(T_s - T_E), \quad (8.1.35)$$
where $\alpha$ – coefficient of heat exchange (heat transfer). It characterizes the heat exchange rate between the body surface and environment. $\alpha$ depends on the velocity of the fluid flow and its temperature, varies along the body surface in the direction of movement. As an approximation of the heat transfer coefficient can be considered constant and independent of the temperature and the same for the entire surface of the body.

According to the law of energy conservation the amount of heat which is removed per unit area per unit time due to heat transfer, must equal the amount of heat applied to one surface by heat conduction from the internal volume of the body:

$$\alpha T_s - T_E = -\lambda \frac{\partial T}{\partial n} S,$$

(8.1.36)

where $n$ – normal to the surface of the body.

In non-stationary processes heat exchange between the surface of the body and the fluid flow $\alpha$ will depend not only on the velocity of the medium and its physical properties, but on the thermophysical properties of the body, and will continuously change over time. Therefore, the ratio $q_s = \alpha(T_s - T_E)$ is used only in special problems of unsteady convective heat transfer.

Boundary conditions of the 3rd kind can be used when considering the heating or cooling body by radiative heat transfer. According to the Stefan-Boltzmann law radiative heat flux between two surfaces:

$$q_s \tau = \sigma^* T_S^4 \tau - T^4,$$

(8.1.37)

where $\sigma^*$ – the emissivity of the surfaces; $T$ – the absolute temperature of the surface of the heat perceiving body. At a low temperature difference $T_S^4 \tau - T^4$ can be written:

$$q_s \tau = \sigma^* T_S^2 \tau - T^2 T_S^2 \tau + T^2 = \alpha T S \tau - T,$$

(8.1.38)

where $\alpha(T)$ – radiative heat transfer coefficient.

$$\alpha \tau = \sigma^* T_S^2 \tau + T^2 T_S \tau + T = \sigma^* b.$$

(8.1.39)

Equation (8.1.38) is an expression of Newton law of cooling or heating the body. Although the relation (8.1.38) is similar to the equation of the law of the convective heat transfer at a constant heat flux, its physical meaning is different. Radiative heat transfer coefficient depends on the temperature and the properties of the surfaces of bodies.
involved in the radiative heat transfer. If temperature $T_s(\tau)$ changes slightly, the coefficient $\alpha(T)$ can be considered as approximately constant.

If the ambient temperature (air) $T_E$ and temperature of the heat perceiving body $T$ are identical, and the absorption coefficient is very small medium, instead $T$ you can write $T_E$. In this case,

$$q_s \tau = \alpha \ T_s \ \tau - T_E.$$  \hspace{1cm} (8.1.40)

Coefficient $\alpha$ is the total heat transfer coefficient:

$$\alpha = \alpha_c + \alpha \ T.$$  \hspace{1cm} (8.1.41)

According to the law of conservation of energy:

$$q_s \ \tau = \alpha \ T_s \ \tau - T_E \ \tau = -\lambda \ \frac{\partial \tau}{\partial n} \ S, \quad (8.1.42)$$

where for generality in the formulation of the problem $T_E$ is variable, and the coefficient of heat transfer is considered to be approximately constant: $\alpha(T)= \alpha = \text{const}$.

Usually boundary conditions of 3rd kind can be written as:

$$\lambda \ \frac{\partial \tau}{\partial n} \ S + \alpha \ T_s \ \tau - T_E \ \tau = 0.$$ \hspace{1cm} (8.1.43)

From the boundary conditions of 3rd kind, as a special case, you can get the boundary conditions of 1st kind. If $\frac{\alpha}{\lambda} \rightarrow \infty$ (coefficient $\alpha \rightarrow \infty$ or thermal conductivity coefficients small $\lambda \rightarrow 0$), then

$$T_s \ \tau = T_E \ \tau = \lim_{\lambda \rightarrow \infty} \ \frac{1}{\alpha} \ \frac{\partial \tau}{\partial n} \ S = 0,$$ \hspace{1cm} (8.1.44)

whence

$$T_s \ \tau = T_E \ \tau.$$ \hspace{1cm} (8.1.45)

I.e. surface temperature of the heat perceiving body is equal to the ambient temperature.
8.2. Differential equations of convective heat transfer

Convection – is the heat transfer in liquids and gases flow of matter. Thus, convection is a simultaneous heat and mass transfer, i.e. heat and mass transfer process. Heat transfer between the surface of the body and the environment can occur under the influence of external forces due to the density difference of the different medium volumes caused by local heating in the field of action of the gravity forces. In the first case, the forced convection takes place, in the second case – a free, natural convection.

For convective heat exchange the heat transfer is carried out simultaneously by convection and thermal conductivity. The heat flux density in this case is:

\[ q = -\lambda \nabla T = \rho h, \]  

(8.2.1)

where \( \rho \) – flow rate; \( h \) – specific enthalpy.

The intensity of the convective heat transfer depends on coefficient of thermal conductivity, specific heat, density, coefficient of viscosity. These values are a function of state parameters (temperature and pressure).

Effect of viscosity on the convective heat transfer process is that during the flow of liquid or gas, having a viscosity, the presence of internal friction leads to dissipation (scattering) of energy. Part of kinetic energy is converted into heat and causes the heating fluid. If the viscosity of the medium or its speed is low, the heating will be negligible.

The value of the compressibility of the medium affects on heat transfer. Isothermal compressibility or compression rate at a constant temperature is a quantity equal to the relative change in density of the medium when the pressure:

\[ \varepsilon = \frac{1}{\rho} \frac{\partial \rho}{\partial p}. \]  

(8.2.2)

For the dropping liquids the isothermal compressibility is very small (for water \( \varepsilon = 5 \times 10^{-10} \text{ Pa}^{-1} \)). Therefore, it is neglected. For normal air isothermal compressibility is more than 20000 times higher than the compressibility of water.

For a significant compression of the gas a significant change in pressure is necessary. If during the gas flow the pressure differences that are small compared to its absolute pressure arisen, the small change in volume obtained, and these gas streams may be considered incompressible. Significant changes in pressure occur at high flow rates. We will consider the heat transfer of incompressible fluids (gases). If the gas velocity is less than \( \frac{1}{4} \) the speed of sound in the medium, the gas can be considered as incompressible.
For convective heat transfer the medium thermal expansion is very important. Expansion coefficient at constant pressure is:

$$\beta = \frac{1}{\nu} \frac{\partial V}{\partial T}, \quad \text{or} \quad \beta = \frac{1}{\rho} \frac{\partial \rho}{\partial T}. \quad (8.2.3)$$

Thermal expansion coefficient is equal to the relative change in volume when changing the temperature by 1 degree.

In an irregular heated environment due to thermal expansion the nonuniform density field occurs, which may lead to a free convection.

In the theoretical analysis of convective heat transfer for simplicity, we assume that the physical properties of the medium are constant in the investigated temperature range.

From equation (8.2.1) it follows that the heat flux density at any point in the medium for each point in time is uniquely determined if known temperature field, the specific enthalpy and the speed.

The relationship between temperature and enthalpy for incompressible media ($\rho = \text{const}$) with sufficient accuracy can be represented as:

$$dh = c_p dT \quad \text{and} \quad h = c_p T. \quad (8.2.4)$$

These equations enable us to establish the relationship between temperature field and the field of enthalpy. To find analytically the temperature field (enthalpy) and speed and determine the $q$, it is necessary to have the appropriate equations.

### 8.2.1. The energy equation

Let us derive a differential equation that describes the temperature field in a moving medium. In the derivation, we assume that the medium is homogeneous and isotropic, its physical parameters are constant, and the strain energy is small compared to the change in internal energy.

We single out in a flow the elementary parallel piped with edges $dx$, $dy$ and $dz$ fixed relatively to coordinate system (Fig. 8.1.1).

Through the faces of the parallelepiped heat is transferred by convection and conduction, and in general in this volume the heat of the internal heat sources can be released. The energy equation for the isobaric process:
\[
\rho \frac{\partial h}{\partial \tau} = -\text{div} \vec{q} + \omega_v. \tag{8.2.5}
\]

where

\[
\text{div} \vec{q} \equiv \frac{\partial q_x}{\partial x} + \frac{\partial q_y}{\partial y} + \frac{\partial q_z}{\partial z}. \tag{8.2.6}
\]

In accordance with the equation (8.2.1), the projections of the heat flow density \( \vec{q} \) on the axes OX, OY and OZ are:

\[
q_x = -\lambda \frac{\partial T}{\partial x} + \rho \nu_x h
\]
\[
q_y = -\lambda \frac{\partial T}{\partial y} + \rho \nu_y h. \tag{8.2.7}
\]
\[
q_z = -\lambda \frac{\partial T}{\partial z} + \rho \nu_z h
\]

Substituting the values of \( q_x, q_y \) and \( q_z \) in equation (8.2.5), we obtain:

\[
\rho \frac{\partial h}{\partial \tau} = \lambda \left( \frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} \right) - \rho \left( \nu_x \frac{\partial h}{\partial x} + \nu_y \frac{\partial h}{\partial y} + \nu_z \frac{\partial h}{\partial z} \right) - \rho h \left( \frac{\partial \nu_x}{\partial x} + \frac{\partial \nu_y}{\partial y} + \frac{\partial \nu_z}{\partial z} \right) + \omega_v. \tag{8.2.8}
\]

For incompressible media (\( \rho = \text{const} \)):

\[
\text{div} \vec{\nu} = \frac{\partial \nu_x}{\partial x} + \frac{\partial \nu_y}{\partial y} + \frac{\partial \nu_z}{\partial z} = 0, \tag{8.2.9}
\]

then

\[
\frac{\partial h}{\partial \tau} + \nu_x \frac{\partial h}{\partial x} + \nu_y \frac{\partial h}{\partial y} + \nu_z \frac{\partial h}{\partial z} = \lambda \left( \frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} \right) + \frac{\omega_v}{\rho} \tag{8.2.10}
\]

or
\[
\frac{\partial T}{\partial \tau} + v_x \frac{\partial T}{\partial x} + v_y \frac{\partial T}{\partial y} + v_z \frac{\partial T}{\partial z} = a \left( \frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} \right) + \frac{\omega_v}{\rho c_p}. \tag{8.2.11}
\]

Equations (8.2.10) and (8.2.11) are the energy equation describing the temperature distribution inside a moving medium.

Polynomial on the left-hand side of equation (8.2.11) is the total derivative of the temperature at the time:

\[
\frac{dT}{d\tau} = \frac{\partial T}{\partial \tau} + \frac{\partial T}{\partial x} \frac{dx}{d\tau} + \frac{\partial T}{\partial y} \frac{dy}{d\tau} + \frac{\partial T}{\partial z} \frac{dz}{d\tau}, \tag{8.2.12}
\]

where \( \frac{dx}{d\tau}, \frac{dy}{d\tau} \) and \( \frac{dz}{d\tau} \) have the meaning of the velocity components \( v_x, v_y \) and \( v_z \). Here \( \frac{\partial T}{\partial \tau} \) characterizes the change in temperature with time at any point in the medium, i.e. is a local change of \( T \); value \( v_x \frac{\partial T}{\partial x} + v_y \frac{\partial T}{\partial y} + v_z \frac{\partial T}{\partial z} \) characterizes the change in temperature during the transition from one point to another, i.e. is a convective changing of \( T \).

The energy equation can be written as:

\[
\frac{dT}{d\tau} = a\nabla^2 T + \frac{\omega_v}{\rho c_p}. \tag{8.2.13}
\]

If \( v_x = v_y = v_z = 0 \), energy equation is transformed into the equation of heat conductivity.

At steady-state convective heat transfer processes \( \frac{\partial T}{\partial \tau} = 0 \). Equation (8.2.11) is further simplified if the temperature changes by only one or two coordinates. In the case of one-dimensional steady-state temperature field all derivatives for \( \tau, y \) and \( z \) equal to zero.

As follows from the equation (8.2.11), the temperature field in the moving medium depends on the velocity components \( v_x, v_y \) and \( v_z \). To make a closed system of equations, you must add the equations that would describe the change in velocity over time and space. Such equations are the equations of motion.
8.2.2. **The equations of motion**

The derivation of the differential equations of motion of a viscous environment requires cumbersome mathematical calculations. In this regard a simplified derivation for the case of one-dimensional flow of an incompressible medium will be given. This conclusion is not strict; its main advantage lies in the clarity. For the three-dimensional equation of motion will be given without proof. The equations of motion are discussed in the course of hydrodynamics.

We distinguish in a viscous medium elementary volume with dimensions of ribs $dx$, $dy$ and $dz$ (Fig. 8.2.1). The rate changes in the flow only in the direction of the axis $y$, the rate variation law is arbitrary.

![Fig. 8.2.1 To the conclusion of the differential equation of the medium motion](image)

Derivation of the equation of motion based on Newton second law: force acting on the volume of the medium is equal to mass times acceleration. Three forces are acting on the fluid element under consideration: gravity, the resultant of the pressure forces and the resultant friction. We find the projection of these forces on the axis OX.

Gravity $df_1$ attached to the centre of gravity of the element. Its projection on the axis OX is the product of the projection of the gravitational acceleration $g$ on the mass of the element:

$$df_1 = \rho g \, dx \, V .$$  \hspace{1cm} (8.2.14)

The resultant pressure force $df_2$ defined as follows. If the upper bound of the pressure medium power element is $p$, then the force $pdydz$ is acting on the side $dydz$.

On the bottom face the pressure up to the second term of the Taylor expansion is $p + \frac{dp}{dx} \, dx$, and on this side the force $-\left( p + \frac{dp}{dx} \right) dydz$ is acting. Here, the minus sign
indicates that the force acts in the opposite direction of fluid flow. The resultant of the pressure forces is equal to the algebraic sum:

\[ df_2 = -\frac{dp}{dx} dV . \]  

(8.2.15)

The resultant friction \( df_3 \) determined from the following considerations. Since speed changes only in the direction of the axis \( OY \), the friction force occurs on the lateral faces of the medium element. Near the left side velocity of medium particles is less than in the element itself, so there is in a cross-section \( y \) friction force directed against the motion and is \( \sigma dx dz \). Near the right side, on the contrary, the velocity of medium particles larger than in the element itself, however sectional \( y + dy \) frictional force is directed in the direction of motion. The resultant of these forces is the algebraic sum:

\[ df_3 = \left( \sigma + \frac{d\sigma}{dy} \right) dx dz - \sigma dx dz = \frac{d\sigma}{dy} dV . \]  

(8.2.16)

Substituting \( \sigma = \eta \frac{d\upsilon_x}{dy} \), we obtain:

\[ df_3 = \eta \frac{d^2\upsilon_x}{dy^2} dV . \]  

(8.2.17)

Summing \( df_1, df_2 \) and \( df_3 \), obtain a projection on the axis \( OX \) of the resultant of all the forces applied to the volume:

\[ df = \left( \rho g_x - \frac{dp}{dx} + \eta \frac{d^2\upsilon_x}{dy^2} \right) dV . \]  

(8.2.18)

According to a second law of mechanics, the resultant is equal to the product of element mass on its acceleration \( \frac{d\upsilon_x}{d\tau} \) and taking into account the inertial forces:

\[ df = \rho \frac{d\upsilon_x}{d\tau} dV . \]  

(8.2.19)
Comparing equations (8.2.18) and (8.2.19) and making cuts, we finally obtain the equation of motion along the axis OX:

\[ \rho \frac{d\upsilon_x}{d\tau} = \rho g_x - \frac{dp}{dx} + \mu \frac{d^2\upsilon_x}{dy^2}. \tag{8.2.20} \]

The motion of the fluid becomes more complicated if the speed changes in three directions. In general, the three-dimensional motion of an incompressible medium with constant physical parameters, the velocity field is described by the three equations of motion, each respectively in projections of forces on the axis OX, OY and OZ:

for OX:

\[ \rho \frac{d\upsilon_x}{d\tau} = \rho g_x - \frac{\partial p}{\partial x} + \eta \left( \frac{\partial^2 \upsilon_x}{\partial x^2} + \frac{\partial^2 \upsilon_x}{\partial y^2} + \frac{\partial^2 \upsilon_x}{\partial z^2} \right), \tag{8.2.21} \]

for OY:

\[ \rho \frac{d\upsilon_y}{d\tau} = \rho g_y - \frac{\partial p}{\partial y} + \eta \left( \frac{\partial^2 \upsilon_y}{\partial x^2} + \frac{\partial^2 \upsilon_y}{\partial y^2} + \frac{\partial^2 \upsilon_y}{\partial z^2} \right), \tag{8.2.22} \]

for OZ:

\[ \rho \frac{d\upsilon_z}{d\tau} = \rho g_z - \frac{\partial p}{\partial z} + \eta \left( \frac{\partial^2 \upsilon_z}{\partial x^2} + \frac{\partial^2 \upsilon_z}{\partial y^2} + \frac{\partial^2 \upsilon_z}{\partial z^2} \right). \tag{8.2.23} \]

Equations (8.2.21) – (8.2.23) are called the Navier-Stokes equations. In general, the components of the velocity \( \upsilon_x, \upsilon_y \) and \( \upsilon_z \), vary in time and space. A term on the left side of equations (8.2.21) – (8.2.23), is the total derivative of speed by the time.

On the basis of the concept of total derivative, we have:

\[ \frac{d\upsilon_x}{d\tau} = \frac{\partial \upsilon_x}{d\tau} + \upsilon_x \frac{\partial \upsilon_x}{\partial x} + \upsilon_y \frac{\partial \upsilon_x}{\partial y} + \upsilon_z \frac{\partial \upsilon_x}{\partial z}. \tag{8.2.24} \]

Similar equations can be written for the other axes. Derivatives \( \frac{\partial \upsilon_x}{d\tau}, \frac{\partial \upsilon_y}{d\tau} \) and \( \frac{\partial \upsilon_z}{d\tau} \) characterize the change of velocity over time in any point of the medium, i.e. characterize a
local change in velocity, and the remaining three terms on the right sides of equations characterize the change in velocity from point to point.

Using the vector notation, equations (8.2.21) – (8.2.23) can be written as

$$
\rho \frac{d\vec{\mathbf{u}}}{d\tau} = \rho \vec{g} - \nabla p + \eta \nabla^2 \vec{\mathbf{u}}.
$$

Equation of motion (8.2.25) obtained excluding dependence of physical parameters of the environment temperature. In particular, it does not include the temperature dependence of the density. At the same time, free convection of the environment determined by the difference of densities of cold and heated volumes.

We restrict ourselves to taking into account the approximate variability of the density. We use this temperature coefficient of volume expansion $\beta$. We assume that in the given temperature range $\beta$ is a constant, independent of temperature.

From the definition of the temperature coefficient of thermal expansion, it follows that $\beta = \text{const}$:

$$
\beta = \frac{\rho - \rho}{\rho \theta}
$$

where $\rho$ and $\rho_0$ – densities corresponding to the temperatures $T$ and $T_0$; $\theta = T - T_0$; $T_0$ – a fixed temperature (reference point). It follows from (8.2.26), that

$$
\rho = \rho_0 \left(1 - \beta \theta\right).
$$

Substituting the value of the density in the equation of motion (8.2.25), we obtain:

$$
\rho \frac{d\vec{\mathbf{u}}}{d\tau} = \rho_0 \left(1 - \beta \theta\right) \vec{g} - \nabla p + \eta \nabla^2 \vec{\mathbf{u}}.
$$

Consider the term $\rho_0 \left(1 - \beta \theta\right) \vec{g} = \rho_0 \vec{g} - \rho_0 \beta \theta \vec{g}$. It can be interpreted as the sum of gravity $\rho_0 \vec{g}$, taken at a certain density, and lift (Archimedean) force $\rho_0 \beta \theta \vec{g}$. Term $\rho_0 \vec{g}$ can be represented as a gradient of hydrostatic pressure $p_0$ in a stationary medium with a density $\rho_0$. Then, instead of $- \nabla p - \rho_0 \vec{g}$ we can write $-\nabla p_1$, where $p_1 = p - p_0$. When replacing the $p$ by $p_1$ the equation of motion will take into account and a member $\rho_0 \vec{g}$. Dropping the
index 0 for \( \rho \) and index 1 for \( p \), after dividing the left and right sides on \( \rho \) we obtain the following equation of motion:

\[
\frac{d\mathbf{\bar{u}}}{d\tau} = \mathbf{\bar{g}}\beta\theta - \frac{1}{\rho} \nabla p + \nu \nabla^2 \mathbf{u}.
\] (8.2.29)

Since in the equation of motion in addition to \( u_x, u_y, u_z \), \( \theta \) includes still an unknown quantity \( p \), then the system is not closed. You need to add one more equation. Such the equation is a differential equation of continuity.

8.2.3. **Continuity equation**

We distinguish in the flow of the moving medium stationary elementary parallelepiped with sides \( dx, dy \) and \( dz \) and calculate the mass of fluid flowing through it in the direction of the axes OX, OY and OZ during \( d\tau \) (Fig. 8.2.2).

![Fig. 8.2.2 To the derivation of the differential continuity equation](image)

In the direction of the axis OX in the mass of the media

\[
dM_x = \rho u_x dy dz d\tau
\] (8.2.30)

flows into a parallelepiped, where \( \rho u_x \) is the mass of fluid flowing per unit time through a unit cross section. The mass

\[
dM_{x+dx} = \rho u_{x+dx} dy dz d\tau
\] (8.2.31)

flows from the opposite face. Taking only the first two terms of the expansion in a row, we see that the mass \( dM_{x+dx} \), flowing from elementary parallelepiped in axis direction OX, is:
Subtracting (8.2.31) from (8.2.32), we get the excess of the mass of the gas flowing out of the volume element in the direction of the axis OX:

\[
dM_{x+dx} - dM_x = \frac{\partial \rho v_x}{\partial x} dV d\tau.
\]  
(8.2.33)

Likewise, for the directions of the axes OY and OZ we have:

\[
dM_{y+dy} - dM_y = \frac{\partial \rho v_y}{\partial y} dV d\tau,
\]  
(8.2.34)

\[
dM_{z+dz} - dM_z = \frac{\partial \rho v_z}{\partial z} dV d\tau.
\]  
(8.2.35)

Summing (8.2.33), (8.2.34) and (8.2.35), we get full excess of the mass of the gas flowing from the considered volume element in the direction of the three axes. This excess is due to the density of the medium change in volume \(dV\) and equal to mass change of current volume over time \(\frac{\partial \rho}{\partial \tau}\) \(dV d\tau\). Making the reduction of \(dV\) and \(d\tau\) and transferring all terms in the left-hand side of the equation, we finally obtain the differential equation for the elastic fluids:

\[
\frac{\partial \rho}{\partial \tau} + \frac{\partial \rho v_x}{\partial x} + \frac{\partial \rho v_y}{\partial y} + \frac{\partial \rho v_z}{\partial z} = 0.
\]  
(8.2.36)

For incompressible media, assuming \(\rho = \text{const}\), we obtain:

\[
\frac{\partial v_x}{\partial x} + \frac{\partial v_y}{\partial y} + \frac{\partial v_z}{\partial z} = 0. 
\]  
(8.2.37)

or, which is the same

\[
\text{div } \vec{v} = 0.
\]  
(8.2.38)

The continuity equation is the equation of conservation of mass.
Thus, convective heat transfer process in an incompressible medium with a homogeneous constant physical parameters described by a system of differential equations (8.2.1), (8.2.11), (8.2.25 or 8.2.29) and (8.2.37)

8.2.4. Conditions of uniqueness

The resulting differential equations describe the convective heat transfer countless specific processes. To highlight the process in question and define it clearly, we need to attach the conditions of uniqueness to the system of differential equations. Conditions of uniqueness give a mathematical description of all the particular features of the phenomenon, and they consist of:

- Geometric conditions characterizing the shape and dimensions of the body, or a system in which the process takes place;
- Physical conditions that characterize the physical properties of the medium;
- Temporal or initial conditions that characterize the features of the process at the initial time;
- Boundary conditions characterizing features of the process at the fluid boundaries.

In the latter the boundary values of the dependent (unknown) variables or their derivatives should be given. For example, for any time the distribution of temperatures or heat fluxes on the surface of the body (in the simplest case $T_E = \text{const}$) or $q_E = -\lambda \left( \frac{\partial T}{\partial n} \right)_{n=0} = \text{const}$; distribution of temperature and velocity in the medium at the inlet channel or at a great distance from the surface in heat; speed values on the wall, etc. are set. Obviously, depending on the setting of the boundary conditions and the other conditions, the results of solutions (integration), as represented by formulas or numerical values, can be varied

The system of differential equations in conjunction with the terms of uniqueness is a mathematical formulation of the boundary value problem.

Physical analysis of convective heat transfer shows that in some cases the mathematical formulation of the problem can be simplified without introducing significant errors. For example, the mathematical formulation can be simplified by using the concept of the boundary layer. Because of the complexity of convective heat transfer in its study the methods of experimental research are widely used. As a result of the experiment the
synthesized information about the process is obtained, the influence of individual factors is not always easy to distinguish. The theory of similarity helps to overcome these difficulties. The basis of the similarity theory is a mathematical formulation of the boundary value problem.

8.2.5. **The hydrodynamic and thermal boundary layers**

Let us consider features of the flow and heat transfer in the boundary layer of the liquid.

8.2.5.1. **Terms of «sticking»**

In the hydrodynamics of viscous fluid there is the hypothesis that the fluid particles directly adjacent to the solid body are adsorbed by it and adhere to its surface, i.e. their velocity is equal to of the body (if the body is stationary – it is equal to zero.) This layer of «sticking» fluid should be considered as an infinitely thin layer. The vanishing of the fluid velocity at the wall is executed as long as the gas can be considered as a continuous medium. With increasing dilution the interaction of the gas with the wall weakens and the low-density gas near the wall starts to slip. The degree of flow dilution is characterized by the Knudsen parameter representing the ratio of the mean free path of the gas molecules to the characteristic size of the solid body (for example, the diameter of the pipe). If \( \frac{\langle l \rangle}{l_0} < 0.001 \) then the gas cannot be considered as a continuous medium, for which the condition of adhesion is performed.

8.2.5.2. **The equation of heat transfer**

Since the solid surface is a thin layer of stationary fluid, the heat flux density at the wall can be determined by Fourier equation:

\[
q_n = -\lambda \frac{\partial T}{\partial n}_n ,
\]

(8.2.39)

where \( n \) – normal to the surface of the body. Thus, if the temperature field is known, the heat flux can be calculated, without recourse to the Newton-Richman law. From the known temperature field the heat transfer coefficient can be determined:

\[
\alpha = \frac{q_n}{T_S - T_E} = -\frac{\lambda}{T_S - T_E} \frac{\partial T}{\partial n}_n .
\]

(8.2.40)
From the condition of zero relative velocity of the liquid on surface of the body other important findings followed by, providing determination of the temperature, and hence, the determination of the heat flux and heat transfer coefficient.

8.2.5.3. Hydrodynamic boundary layer

Let us consider longitudinal flow over a flat surface of the body of an unbounded environment stream. The speed and temperature of the incoming flow is constant and equal \( u_0 \) and \( T_0 \). In contact of the medium with the body surface in the field around the plate due to the action of viscous forces a thin layer of the medium inhibited, within which the velocity varies from zero at the surface to the undisturbed flow rate (far from the body). This layer is called the hydrodynamic boundary layer. The greater the distance from the front edge of the plate, the thicker boundary layer, as the effect of viscosity as it moves along the body of the medium penetrates farther into the undisturbed flow.

For fluid flow inside the boundary layer the condition \( \frac{\partial v_x}{\partial y} \neq 0 \) is true, and outside the boundary layer and at its outer edge \( \frac{\partial v_x}{\partial y} = 0 \) and \( v_x = u_0 \).

The concepts of «boundary layer thickness» and «outer limit of the boundary layer» are conditional, as there is not a sharp transition from the boundary layer to the current layer. By the boundary layer thickness the distance from the wall on which the velocity is different from a flow rate away from the body for a certain small value preassigned is meant.

Thus, when flowing around the body liquid stream as though is separated into two parts: boundary layer and the external flow. In the external flow the inertial forces (convective heat transfer) are dominated, viscous forces do not appear here. In the boundary layer viscous forces and inertial forces are comparable. Then we can write the following system of differential equations describing the steady-state velocity field in the washing of a flat plate, infinite in the direction of the axis OZ.

The equations of motion:

\[
\begin{align*}
  u_x \frac{\partial v_x}{\partial x} + u_y \frac{\partial v_x}{\partial y} &= \nu \left( \frac{\partial^2 v_x}{\partial x^2} + \frac{\partial^2 v_y}{\partial y^2} - \frac{1}{\rho} \frac{\partial p}{\partial x} \right), \\
  u_x \frac{\partial v_y}{\partial x} + u_y \frac{\partial v_y}{\partial y} &= \nu \left( \frac{\partial^2 v_y}{\partial x^2} + \frac{\partial^2 v_y}{\partial y^2} - \frac{1}{\rho} \frac{\partial p}{\partial y} \right).
\end{align*}
\] (8.2.41)

Continuity equation:
Due to the small thickness of the boundary layer we accept that the pressure across it is not changed, i.e. $\frac{\partial p}{\partial y} = 0$. With a washing a flat surface by unlimited flow when the external flow velocity is constant and equal to $u_0$, from Bernoulli's equation it follows that in the external flow and the pressure does not change, that is $\frac{\partial p}{\partial x} = 0$. For the boundary layer, this condition is also true. For the boundary layer $\delta \ll l$ (thickness many times smaller than the dimensions of the plate) whereas

$$\frac{\partial^2 u_x}{\partial y^2} \gg \frac{\partial^2 u_x}{\partial x^2}. \quad (8.2.44)$$

The last of the derivative can be neglected. For the boundary layer equation (8.2.42) can be omitted since the members are small in comparison with those in (8.2.41). Then for the plane steady flow of viscous fluid in the boundary layer at the flat surface of the following relations are valid:

$$\nu_x \frac{\partial u_x}{\partial x} + \nu_y \frac{\partial u_x}{\partial y} = \nu \frac{\partial^2 u_x}{\partial y^2}, \quad (8.2.45)$$

$$\frac{\partial u_x}{\partial x} + \frac{\partial u_y}{\partial y} = 0. \quad (8.2.46)$$

Thus, the theory of the boundary layer is a method of simplifying the mathematical formulation of the boundary value problem and the related possibility of solving.

8.2.5.4. Thermal boundary layer

The thermal boundary layer − is a layer of liquid near the wall, within which the temperature changes from a value equal to the wall temperature, to a value equal to the ambient temperature away from the body. For the region inside of the thermal boundary layer we have the condition $\frac{\partial T}{\partial y} = 0$, and at the outer edge and beyond $\frac{\partial T}{\partial y} = 0$ and $T = T_0$. Thus, all changes in temperature are concentrated in a relatively thin layer immediately adjacent to the body surface. Thicknesses of hydrodynamic and thermal boundary layers are generally don’t coincide − it depends on the kind of the medium and some parameters of flow and heat transfer. Due to the small thickness of the thermal boundary layer thermal conductivity along the layer compared with the transverse transfer of heat is negligible, that is, we put:
Then, for the considered case the energy equation takes the form:

\[
\frac{\partial^2 \tau}{\partial x^2} = 0, \tag{8.2.47}
\]

\[
\frac{\partial^2 \tau}{\partial x^2} \ll \frac{\partial^2 \tau}{\partial y^2}, \quad \text{as} \delta' \ll l. \tag{8.2.48}
\]

The system of differential equations (8.2.45), (8.2.46), (8.2.49) obtained for a stationary process of convective heat transfer surface with the medium in the case of forced convection medium has constant physical properties of the gas are no internal heat sources, heat generation of friction is negligible. Heat emission depends strongly on the flow regime. The resulting system of differential equations (8.2.45), (8.2.46), (8.2.49) describes the heat transfer in the laminar boundary layer.

The boundary layer in case of a free convection caused by a density difference and less volume of the heated medium is constructed uniquely. The definition of the boundary layers is also valid for free movement. However, in many instances, the speed away from the body surface is zero. In this case, the thermal and hydrodynamic thickness of the layers may be different. In the free thermal movement in the differential equation of motion must be added a term \( g\beta \theta \). In this case, the velocity field is closely related to the temperature field.

8.2.5.5. Similarity of the convective heat transfer processes

Convective heat transfer is described by a system of differential equations and conditions for the uniqueness of a large number of variables.

When introduced into the equation of dimensionless complexes number of units under the sign of the desired function is formally reduced, which simplifies the study of physical processes. In addition, the new dimensionless variables reflect the effect of not only individual factors, but also their set, which makes it easier to determine the physical connection to the process under study.

Similarity theory establishes the conditions under which the laboratory results can be extended to other phenomena, like under consideration. In view of the similarity theory is primarily a theoretical basis of the experiment, as well as an important tool of theoretical studies. Although the methods of the theory of similarity view of the desired function
cannot be determined, this theory makes it easier in some cases, process analysis and description of the results.

For practical use of the findings of the similarity theory we must be able to lead to dimensionless form of the mathematical descriptions of the processes under study.

There are several methods for performing this operation. We'll use one of them – a method of large-scale transformations.

Let the surface of the solid is washed by incompressible medium, temperature and velocity is away from the body are constant and equal to \( T_0 \) and \( u_0 \). The body size \( l_0 \) is set. The surface temperature of the body is \( T_s \). For definiteness, assume that \( T_s > T_0 \). We assume that the physical parameters of the medium are constant (we take into account only the lift force generated by the density depending on temperature).

The heat of friction is not taken into account. The process under consideration is stationary.

Arrange the axes so that the OY axis normal to the surface of the body and the axis OX is directed along and the body and is vertical, wherein \( g_x = g \), and the projections of the vector of gravity (or lift) on the axis OY and OZ will be zero (\( g_y = g_z = 0 \)) (Fig. 8.2.3).

![Fig. 8.2.3. To setting the boundary value problem of convective heat transfer](image)

The size of the body along the axis OZ is much more \( l_0 \). With the assumed conditions the temperature and velocity fields can be described by differential equations in the approximation of the boundary layer. We take into additionally account lift \( pg\beta \theta \),
considering it is commensurate with viscous term $\eta \frac{\partial^2 u_x}{\partial y^2}$. We also introduce the notation 
$\theta = T - T_0$, where $T$ – medium temperature ($dT = d\theta$ as $T_0 = \text{const}$).

The energy equation

$$\nu_x \frac{\partial \theta}{\partial x} + \nu_y \frac{\partial \theta}{\partial y} = a \frac{\partial^2 \theta}{\partial y^2}, \tag{8.2.50}$$

equation of motion

$$\nu_x \frac{\partial \nu_x}{\partial x} + \nu_y \frac{\partial \nu_y}{\partial y} = \nu \frac{\partial^2 \nu_x}{\partial y^2} - g\beta \theta, \tag{8.2.51}$$

continuity equation

$$\frac{\partial \nu_x}{\partial x} + \frac{\partial \nu_y}{\partial y} = 0. \tag{8.2.52}$$

Let us write the boundary conditions:

1) away from the body $y = \infty$
   \[
   \begin{align*}
   \theta &= \theta_0 = 0 \\
   \nu_x &= \nu_0 \\
   \nu_y &= 0
   \end{align*}
   \]

2) on the body surface $y = 0$, $0 \leq x \leq l$, $-\infty \leq z \leq +\infty$
   \[
   \begin{align*}
   \theta &= \theta_s = T_s - T_0 = \text{const} \\
   \nu_x &= \nu_y = \nu_z = 0
   \end{align*}
   \]

In the equations and conditions of uniqueness we can distinguished three kinds of variables: independent variables – the coordinates $x, y$; dependent variables – $\theta, \nu_x$ and $\nu_y$ (dependent variables uniquely determined by the values of the independent variables, if there are given quantities entering into terms of uniqueness); constants – $\nu_0, T_0, l_0, \theta_s, \nu, a, g\beta$ and other (they are given by the terms of the uniqueness and for a specific task are constant and independent of other variables, but from task to task they are subject to
change, these quantities are called constant because they are not a function of the independent variables).

Thus, the unknown variables $\theta$, $\nu_x$, and $\nu_y$ depend on a large number of variables: they are a function of the independent variable and fixed quantities in terms of uniqueness.

The values contained in the equations and conditions of uniqueness can be grouped into complexes. The number of dimensionless complexes will be less than the number of dimensional quantities.

To bring to a dimensionless form we choose the scale of the cast. As the scale it is convenient to take continuous quantities entering into terms of uniqueness. For linear units we choose a distinctive size, such as the length of the heat transfer surface $l_0$, velocity $\nu_0$, temperature $\theta_0$.

We denote the dimensionless quantities:

$$
X = \frac{x}{l_0}; \quad Y = \frac{y}{l_0}; \quad V_x = \frac{\nu_x}{\nu_0}; \quad V_y = \frac{\nu_y}{\nu_0}; \quad \Theta = \frac{\theta}{\theta_E}.
$$

(8.2.54)

then

$$
x = l_0 X; \quad y = l_0 Y; \quad \nu_x = \nu_0 V_x; \quad \nu_y = \nu_0 V_y; \quad \theta = \theta_E \Theta.
$$

(8.2.55)

Substitute in equations (8.2.50) – (8.2.52) the values according to the equations (8.2.55). Transform the energy equation. Since, for example

$$
\frac{\partial^2 \theta}{\partial y^2} = \frac{\partial}{\partial \frac{Y}{l_0}} \left( \frac{\partial \theta E \Theta}{\partial \frac{Y}{l_0}} \right) = \frac{\theta_E}{l_0^2} \frac{\partial^2 \Theta}{\partial Y^2},
$$

(8.2.56)

then the result of the substitution of (8.2.55) by multiplying the left and right sides of the energy equation on $\frac{l_0^2}{a}$ we will have:

$$
\frac{\nu_0}{a} \frac{l_0}{V_x} \frac{\partial \Theta}{\partial X} + \frac{\nu_0}{a} \frac{l_0}{V_y} \frac{\partial \Theta}{\partial Y} = \frac{\partial^2 \Theta}{\partial Y^2}.
$$

(8.2.57)
Similarly, we transform the equation of motion. After substitution of (8.2.55) into the equation of motion multiply it by \( l_0^2/\nu u_0 \). As a result, we obtain:

\[
\frac{\nu l_0}{v} \left( V_x \frac{\partial V_x}{\partial X} + V_y \frac{\partial V_x}{\partial Y} \right) = \frac{\partial^2 V_x}{\partial Y^2} + \frac{g \beta \theta l_0^2}{\nu u_0} \Theta. \tag{8.2.58}
\]

We make the following transformation of the complex included in the last equation:

\[
\frac{g \beta \theta l_0^2}{\nu u_0} \Theta = \frac{g \beta \theta l_0^3}{\nu^2} \frac{\nu}{u_0 l_0} \Theta. \tag{8.2.59}
\]

Given these changes, we finally obtain:

\[
\frac{\nu l_0}{v} \left( V_x \frac{\partial V_x}{\partial X} + V_y \frac{\partial V_x}{\partial Y} \right) = \frac{\partial^2 V_x}{\partial Y^2} + \frac{g \beta \theta l_0^3}{\nu^2} \frac{\nu}{u_0 l_0} \Theta. \tag{8.2.60}
\]

After the transformation the continuity equation we obtain:

\[
\frac{\nu}{l_0} \left( \frac{\partial V_x}{\partial X} + \frac{\partial V_y}{\partial Y} \right) = 0 \tag{8.2.61}
\]

or because \( \frac{\nu}{l_0} \) is not a zero,

\[
\frac{\partial V_x}{\partial X} + \frac{\partial V_y}{\partial Y} = 0. \tag{8.2.62}
\]

The resulting dimensionless form the boundary conditions, we obtain:
1) away from the body \( Y = \infty \)
\[
\begin{align*}
\Theta &= \Theta_0 = 0 \\
V_x &= 1 \\
V_y &= 0
\end{align*}
\]
\[ (8.2.63) \]

2) on the body surface \( Y = 0, \, 0 \leq X \leq 1 \)
\[
\begin{align*}
\Theta &= \Theta_s = 1 \\
V_x &= V_y = 0
\end{align*}
\]

From (8.2.63) it follows that despite the fact that the quantities \( \nu_0, \, T_S, \, T_0 \) and other entering into the dimensional boundary conditions can have different numerical values, each of the dimensionless quantities \( \Theta_0, \, \Theta_s, \, \ldots \) in this case has a very specific numeric value.

With the known temperature field the heat transfer coefficient can be determined by the equation
\[
\alpha = -\frac{\lambda}{T_S - T_0} \left( \frac{\partial T}{\partial y} \right)_{y=0}.
\]
\[ (8.2.64) \]

Quoting an expression to a dimensionless form, we obtain:
\[
\frac{\alpha l_0}{\lambda} = -\left( \frac{\partial \Theta}{\partial Y} \right)_{Y=0}.
\]
\[ (8.2.65) \]

In addition to the dimensionless quantities \( \Theta, \, V_x, \, V_y \) and the dimensionless coordinate composed of homogeneous physical quantities, equations also includes the dimensionless complexes consisting of heterogeneous physical quantities: \( \frac{\alpha l_0}{\lambda}, \, \frac{\nu l_0}{v}, \, \frac{\nu l_0}{a}, \, \frac{g \beta_0 l^3}{v^2} \).

These complexes, called the similarity numbers are assigned to the names of scientists who have made significant contributions to the development of hydrodynamics and heat transfer.

The first of these dimensionless complexes represent:
and called the Nusselt number or the dimensionless heat transfer coefficient. Nusselt number characterizes the heat transfer at the border wall-medium. In the problems of convective heat transfer Nu number is usually unknown quantity, since it includes a defined value $\alpha$.

Dimensionless complex:

$$ Nu = \frac{al_0}{\lambda} $$

(8.2.66)

and called the Reynolds number. It characterizes the ratio of inertial forces and viscous forces.

Dimensionless complex:

$$ Re = \frac{\nu_0 l_0}{\nu} $$

(8.2.67)

called the Reynolds number. It characterizes the ratio of inertial forces and viscous forces.

The third dimensionless complex denote

$$ Pe = \frac{\nu_0 l_0}{a} $$

(8.2.68)

and called the Peclet number. It can be transformed as follows:

$$ \frac{\nu_0 l_0}{a} = \frac{\rho c_p \nu_0 \theta}{\frac{\lambda}{\theta} \frac{l_0}{l_0}} $$

(8.2.69)

Here the numerator characterizes the heat carried by convection, and the denominator – the heat carried by the thermal conductivity.

Dimensionless complex:

$$ Gr = \frac{g \beta \theta l_0^3}{v^2} $$

(8.2.70)

is called Grashof number. It characterizes the lifting force generated in the environment due to the density difference. Since during the derivation of equation of motion it was assumed that $\beta \theta = \frac{\rho_0 - \rho}{\rho_0}$, instead $Gr$ you can write it more common modification – the number of Archimedes:
\[ Ar = \frac{g l_0^2}{\nu^2} \frac{\rho_0 - \rho}{\rho_0} . \]  

(8.2.71)

For a homogeneous medium, provided \( \beta = \text{const} \) Archimedes number identical to the number \( Gr \).

Number \( Pe \), obtained in the reduction to dimensionless form of the energy equation can be written as the product of two dimensionless variables:

\[ Pe = Re Pr = \frac{\nu_0 l_0}{\nu} \frac{\nu}{a} . \]

(8.2.72)

Dimensionless complex \( Pr = \frac{\nu}{a} \) is a new variable called Prandtl number. It can be written as:

\[ Pr = \frac{\nu}{a} = \frac{\mu c_p}{\lambda} . \]

(8.2.73)

Prandtl number can be given a definite physical meaning. The energy equation

\[ \nu_x \frac{\partial T}{\partial x} + \nu_y \frac{dT}{dy} = a \frac{\partial^2 T}{\partial y^2} \]

(8.2.74)

and the equation of motion

\[ \nu_x \frac{\partial u_x}{\partial x} + \nu_y \frac{\partial u_x}{\partial y} = \nu \frac{\partial^2 u_x}{\partial y^2} \]

(8.2.75)

by recording are the same. At \( a = \nu \) calculated temperature and velocity fields will be similar, unless the conditions of uniqueness are similar. The equality \( Pr = 1 \) corresponds for condition \( a = \nu \). Thus, under certain conditions, the Prandtl number can be given a sense of similarity measure of temperature and velocity fields.

Using this notation, the system of dimensionless differential equations can be written as:

\[ Nu = - \frac{\partial \Theta}{\partial y} \bigg|_{y=0} , \]

(8.2.76)

\[ Pe \left( V_x \frac{\partial \Theta}{\partial x} + V_y \frac{\partial \Theta}{\partial y} \right) = \frac{\partial^2 \Theta}{\partial y^2} . \]

(8.2.77)
The system of dimensionless differential equations and dimensionless terms of uniqueness is a mathematical formulation of the problem.

\[
Re \ V_x \frac{\partial v_x}{\partial x} + V_y \frac{\partial v_x}{\partial y} = \frac{\sigma r}{Re} \Theta + \frac{\partial^2 v_x}{\partial y^2}, \tag{8.2.78}
\]

\[
\frac{\partial v_x}{\partial x} + \frac{\partial v_y}{\partial y} = 0. \tag{8.2.79}
\]
9. Experimental methods for determining of thermo physical characteristics

Nina Karbalevich
Belarusian State University, Physical Faculty
Introduction

The limitation of energy causes the growing interest to the problems of production, transmission of energy and energy management.

The intensive development of industry and energy, implementation of new technologies and materials put forward the task to assess the impact of scientific and technological progress, and to prevent or reduce harmful effects on the surrounding world. The problem of changing the current situation of heat nowadays becomes more urgent, because heat is not only a prerequisite to the emergence of different thermal conditions, but also changes the nature of the processes occurring in the air and water, biological systems, environmental conditions in general.

Thermal environment around us is created by a wide variety of heat sources and limiting construction. Measurement of various thermal values characterizing these sources and walling materials is an important task of power, heating and thermal physics. These values are the temperature coefficients of thermal conductivity and thermal diffusivity, heat capacity, heat transfer coefficients.

9.1. Methods for measuring the temperature

A constant and highly accurate temperature control is very important in modern power systems.

This applies to systems of production, transmission and use of not only the heat for which the temperature is one of the main parameters, but also electric power and a variety of control systems in which many components dissipate more heat output, and constant temperature control is important for the maintenance of such systems in an efficient and most efficient condition.

Precise temperature measurements while the implementation of new energy-saving technologies are important for the accurate assessment of their effectiveness.

Determination of the most important characteristics of any thermal process – the temperature – is associated with the concept of thermodynamic equilibrium. Temperature is a physical quantity that characterizes the state of thermodynamic equilibrium of a macroscopic system. It is the same for all parts of an isolated system in thermodynamic equilibrium. In equilibrium, the temperature is proportional to the average kinetic energy of the particles of the system.
Temperature is a key parameter of thermal systems, but its value cannot be measured directly. If one changes the thermal state of the body other than the temperature changes and its other physical characteristics (e.g. size). By quantitative change of these characteristics (volume, electromotive force, and electric resistance) one can assess the change of the body temperature.

Temperature measurement is carried out by highly sensitive sensors: thermocouples, thermometers, thermistors, etc. However, there are situations where the use of the converters mentioned above cannot estimate the temperature field of the object, for example, the temperature distribution in some areas of the details. For measuring the temperature fields of energy equipment and machinery, natural objects and structures infrared sensors, liquid crystals, heat-sensitive inks and films are used.

By the nature of the interaction of temperature measuring instrument and object of study methods of measurement are divided into contact and non-contact.

Contact temperature meters are widely used to measure the temperature of solid, liquid and gaseous media. High accuracy and wide scope of measured temperatures allow using these sensors also in the study of various physical processes and phenomena (e.g. Hall effect, Seebeck effect, and others).

Contact temperature sensors are resistance thermometers, thermocouples, various kinds of thermometers, etc. Non-contact temperature sensors are pyrometers, interferometers, photometers.

9.1.1. **The study of resistance thermometers and their graduation**

Application of resistance thermometers as temperature sensors is based on the dependence of metal (semiconductor) resistance on temperature. These dependencies are of fundamentally different kind for metals and semiconductors.

The conductivity of the crystal is proportional to the concentration of free charge carriers and their mobility. In metals, the concentration of free carriers is almost independent of temperature. However, the intensity of the thermal motion of ions in the crystal lattice increases with the temperature increase (in other words, the number of phonons in the crystal increases). This leads to an increase of the number of conduction electrons collisions with phonons, resulting in the effect when temperature increases, the mobility of carriers is reduced and the resistance of the crystal is increased. Experiments
have shown that the resistivity of metals increases with temperature to a power law. In a relatively narrow temperature range this dependence is close to linear.

In semiconductors (as in metals) the temperature increases, the mobility of carriers is reduced. Unlike metals, in semiconductors the ionization of atoms because of their thermal motion takes place with the emergence of electrons (or holes) of conduction under the temperature increase. For various semiconductors valence electrons ionization energy lies in the range from 0.1 to 5 eV. The carrier concentration in the semiconductor sharply increases exponentially as the temperature rises, resulting in conductivity of semiconductors with increasing temperature increases despite a decrease in mobility.

On the property of metals and alloys to change their electrical resistance as a function of temperature the principle of resistance thermometers operation is established.

Thermal resistance is a conductor included in the circuit that is in a state of heat exchange with the environment. Its resistance is temperature dependent and is determined by the thermal balance between the conductor and its environment. The heat transfer between the conductor and the environment under the study can be carried out by convection, thermal conductivity of the medium, heat conductivity of the conductor and radiation.

The materials used as thermistors must meet the requirements of the stability characteristics of \( R = f \left( T \right) \) and high enough temperature coefficient \( \alpha \), determined by expression

\[
\alpha = \frac{1}{R} \frac{dR}{dT}.
\]  

(9.1.1)

For most of pure metals the temperature coefficient approximately equal to 4.10^{-3} K^{-1} at room temperature. In some cases, the sensitivity of resistance thermometers allows to measure the temperature with a 0.001 K.

The greatest distribution for the production of resistance thermometers transducers received platinum, copper, nickel, iron, bronze, pyrolytic graphite, some alloys and semiconductors made of mixed oxides of different metals. The best material is pure platinum that doesn’t enter into chemical reactions in the wide range of temperatures and consistently maintains the value of the resistivity (about five times greater than that of copper, silver or gold).

Besides platinum, commercially produced also technical copper resistance thermometer with nominal values \( R_0 = 50 \) and 100 Ohm, designed to operate at temperatures from – 50
to +180°C. In this temperature range, the electrical resistance of copper thermometers is defined by equation

\[ R_t = R_\circ (1 + \alpha t), \quad (9.1.2) \]

where \( \alpha \) – temperature coefficient, on average equal to 4.26\times 10^{-3} \text{ K}^{-1}. At higher temperatures the copper is oxidized; the one more disadvantage of copper is also its low resistivity. For temperature measurements \( \sim 300 \text{ °C} \) the nickel is the first replacement of platinum. It has a large resistivity (\( \sim 8 \times 10^{-8} \text{ Ohm} \cdot \text{m} \)) and high temperature coefficient \( \alpha = 6.4 \times 10^{-3} \text{ K}^{-1} \), but at 370 °C nickel undergoes structural transformation and the function \( R = f (T) \) becomes ambiguous. To a temperature of 100 °C iron (\( \alpha = 6.5 \times 10^{-3} \text{ K}^{-1} \)) can be used.

Semiconductor thermistors are made from a mixture of oxides or sulfides of metals or different rare earth elements. Semiconductor thermistors have a very high value of the negative temperature coefficient of resistance, reaching up to \( 4 \times 10^{-2} – 5 \times 10^{-2} \text{ K}^{-1} \). They have low inertia and can be used to study transient thermal processes.

Semiconductor resistance thermometers have a nonlinear dependence of resistance on temperature, which is approximately governed by an exponential law

\[ R_T = A_T \exp \left( -\frac{B}{T} \right), \quad (9.1.3) \]

where \( A_T \) and \( B \) – coefficients depending on the physical properties and the geometry of a semiconductor, with a considerable change in temperature. The coefficient \( A_T \) is a function of temperature \( T \) and may be constant in only small changes in the intervals \( T \). A significant drawback of semiconductor thermistors is the low stability of their characteristics that increases while the temperature increased, so the range of use is limited to temperatures not exceeding 300 °C. The most common semiconductor thermistors are used in cryogenics.

During the calibration of metal and semiconductor thermal resistances the dependence \( R = f (T) \) is determined from experiment and the temperature coefficient is calculated. Calibration of resistance thermometers intended for precise measurements, performed at constant melting and boiling points of pure chemicals.
When measuring the temperature of the medium, the resistance of the thermometer that is in thermal equilibrium with the environment is determined, and the desired temperature is found by means of the calibration curve or table.

9.1.2. **Thermoelectric phenomena. Temperature measurement using thermocouples**

Thermoelectric phenomena can be explained by the fact that the average energy of free electrons is different in different conductors and increases differently with temperature. If there is a temperature drop along the conductor, the electrons at the hot end gain higher speed and energy than at the cold. The result is a flow of electrons from the hot end to the cold one, and the cold end accumulates negative charge while the hot end remains uncompensated positive charge. This accumulation process will continue as long as the potential difference originated in such way will cause the flow of electrons, equal to the primary one, in the opposite direction and due to this the steady-state condition will be established. The algebraic sum of the potential differences in the chain creates one (volume) component of the thermo EMF.

The second (contact) component – a consequence of the temperature dependence of the contact potential difference. Contact potential difference is caused by the difference in the electrons work functions and the concentrations of free electrons in metals listed in the contact. If both of the thermocouple junctions are at the same temperature, the contact potential differences cancel each other. If the junction temperature varies, the contact potential difference contributes to the thermo EMF, which can be comparable or even greater than the bulk thermo EMF.

Another source of thermo EMF is the drag of electrons by phonons. If the solid temperature gradient exists, the number of phonons, moving from the cold end to the hot one will be greater than in the reverse direction. With its directional movement phonons as a result of collisions with electrons will drag them and the negative charge will accumulate at the cold end of the sample (as positive charge will accumulate at the hot one), as long as potential difference originated in a suchway will balances the drag effect. So, this potential difference is an additional component of the thermo EMF, which can be in the tens or hundreds of times greater than the discussed above at low temperatures. The concentration of free electrons in metals is very large ($\sim 10^{22}$ sm$^{-3}$) and doesn’t depend on temperature. The energy and velocity of the electrons is also almost independent of temperature.
Therefore, the thermo EMF of «classic» metal is very small (a few $\mu V/K$). For semiconductors the unit thermo EMF is of the order of $10^2$–$10^3\mu V/K$.

Generally the unit thermo EMF $\alpha = \alpha_0 + \alpha_K + \alpha_P$, where $\alpha_0$— the bulk component of thermo EMF, that is due to the preferential diffusion of charge carriers in a conductor or semiconductor from the heated end to the cold one; $\alpha_K$— contact component of thermo EMF, associated with the temperature dependence of the chemical potential; $\alpha_P$— phonon thermo EMF component.

Closed circuit of conductors, creating current by temperature differences between the contact wires, called a thermoelement or thermocouple. The application of thermocouples as temperature sensors is based on the use of the Seebeck thermoelectric effect which consists in the occurrence of EMF in the circuit of two or more conductors of different materials, if their junctions are at different temperatures. The thermo EMF value is proportional to the temperature difference: $\varepsilon = \alpha \Delta T$.

Metal thermocouples are used for temperature measurement. For this purpose the circuit shown in Fig. 9.1.1 is made. One thermocouple junction, consisting of the conductors 1 and 2 is maintained at a certain constant temperature (e.g. at a temperature of ice melting). The second junction is placed in the environment with the temperature $T$ to be measured. Knowing the thermocouple factor $\alpha$ and measuring the thermo EMF the temperature is calculated by the formula

$$T_a = \frac{\varepsilon + \alpha T_b}{\alpha}.$$ (9.1.4)

![Fig. 9.1.1. Differential thermocouple](image)

Thermoelectric thermometer has significant advantages over other sensors. It is highly sensitive and has low thermal inertia, so it’s applicable in a wide range of temperatures, and allows measuring the temperature of small volumes of media. In addition, it allows the
remote measurements, i.e., the determination of the temperature of the object, which is located at a great distance from the place of measurement.

The heat-insulated melting ice bath is most commonly used to stabilize the temperature of the non-working junction. The melting point of ice (273.16 K) is useful because the various factors effect on its temperature insignificantly. If in the process of calibrating the temperature $T_0$ is maintained on the free ends and the temperature $T$– on operation ends, then the amendment is added to the measured thermo EMF.

For measuring temperatures up to 1100 °C mainly base-metal thermocouples are used, for measuring temperatures from 1100 to1600 °C – thermocouples of platinum-group noble metals, for measuring temperatures over 1600 °C – the various thermocouples made of heat-resistant materials.

From the number of base-metal thermocouples are most often used Chromel-Copel and Chromel-Alumel thermocouples. Thermocouple Ch-A has almost linear characteristic, and thermocouple Ch-C at much higher nonlinearity develops a very high thermo EMF, which explains its widespread use.

Among the base-materials thermocouples there are standard thermocouples, whose characteristics are listed in Table 9.1.1.

Table 9.1.1 Thermocouple characteristics

<table>
<thead>
<tr>
<th>Thermocouple</th>
<th>Thermo EMF (mV/100 °C)</th>
<th>$T$ (°C)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Copper-Copel</td>
<td>4,75</td>
<td>350 – 500</td>
</tr>
<tr>
<td>Iron-Copel</td>
<td>5,75</td>
<td>600 – 800</td>
</tr>
<tr>
<td>Chromel-Copel</td>
<td>6,90</td>
<td>600 – 800</td>
</tr>
<tr>
<td>Chromel-Alumel</td>
<td>4,10</td>
<td>1100 – 1250</td>
</tr>
<tr>
<td>Copper-Constantan</td>
<td>4,0</td>
<td>250 – 400</td>
</tr>
</tbody>
</table>

Noble-metals thermocouples. These thermocouples, belonging to the platinum group, are widespread primarily as exemplary thermocouples and thermocouple for measuring temperatures above 1000 ° C. The main thermocouple of this group is a Platinum Rhodium-Platinum thermocouple, one of which thermo electrodes is a pure platinum and the second one is an alloy (90 % Pt + 10 % Rh). This thermocouple can measure temperatures up to 1600 °C, briefly up to 1400 °C for a long time. Thermocouple PtRh-Pt develops thermo EMF equal to 17 mV at 1600 °C (at the free ends temperature equal to zero).
The thermocouples are made of nobel-metal alloys are suitable for measuring temperatures up to 1700 and 1800 °C (PtRh 30/6 thermocouple).

In the PtRh 30/6 thermocouple positive thermoelectrode is an alloy consisting of 70 % Pt and 30 % Rh, and negative thermoelectrode is an alloy of 94 % Pt and 6 % Rh. The upper limit for the short measured temperature of this thermocouple can take a temperature of 1750 °C. Thermocouples Platinum Rhodium-Platinum Rhodium have a somewhat lower thermo EMF: for example, the PtRh 30/6 thermocouple thermo EMF is equal to 10,82 mV under 1546 °C.

Since the thermo EMF of the thermocouple at room temperature is very low, the thermo stating of its free ends is excessive.

Except PtRh 30/6 thermocouple similar thermocouple with other rhodium content can be used (PtRh 40/10, PtRh 30/15, PtRh 40/20), having a slightly lower EMF than PtRh 30/6 thermocouple, but suitable for measuring a little higher temperatures (up to 1800 – 1850 °C).

9.2. Methods for measuring thermal conductivity and thermal diffusivity

Now in the world the significant attention is given to solving problems related to energy efficiency in industry and everyday life.

Insufficient building envelope heat resistance can serve as a major source of heat loss.

To study the processes of heat and mass transfer through the building envelope it is necessary to have the known values of the coefficients of heat and mass transfer of material layers that comprise the cladding. Accuracy of the heat engineering calculations to a large extent depends on how well the values of construction building materials thermal characteristics are selected.

In the calculations of heat conduction problems through the building envelope, the following coefficients are used: thermal diffusivity $a$, thermal conductivity $\lambda$, heat exchange $\alpha$, specific heat capacity $c$. The numerical values of these factors vary considerably depending on the composition, structure, heat and humid conditions and material conditions of heat exchange of surfaces with the medium.

In the building materials science coefficients of thermal conductivity and thermal diffusivity are taken into account in solving the heat envelope for providing:
Thermal insulation of buildings and structures;
Thermal protection of the surfaces of thermal units and pipelines;
Thermal stability of refractory materials and special trains;
Cold insulation.

Determination of the coefficients of heat conductivity and thermal diffusivity implemented in different ways in different temperature regimes: nonstationary, regular and steady. In this case heat sources may act for the whole time of the experiment (constant power source), a short time (pulse power), or periodically (the method of thermal waves).

Formation of temperature fields and the related possibilities of finding thermal characteristics are determined by the boundary conditions.

If, for example, heated or cooled body is placed in a constant temperature environment, the change in its temperature over time includes three stages: the initial (irregular), regular and stationary. The first stage is characterized by a disorder of the thermal regime and its strong dependence on the initial temperature distribution. The main feature of the second step – the regular mode is that the change in temperature, described by the first term of the series in the solution of the differential equation of heat conduction, does not depend on the initial temperature distribution. The third stage corresponds to the steady state in which the temperature of all the points of the body is the same and equal to the ambient temperature. In the absence of sources of heat in the body stationary phase can be used to calculate the thermal conductivity, as it corresponds to the full temperature equalization in volume of the body. In the presence of sources under the same conditions calculation of thermophysical characteristics can be carried out on any part of the temperature curve.

At a certain value of the Fourier number \( (F_o \geq 0.2) \) corresponding to the regular mode and the specified accuracy, the range can be replaced by its first term.

A characteristic feature of the regular thermal regime is also a uniform rate of cooling (heating) \( m \) for all points of the body and its independence from the initial temperature distribution:

\[
m = \frac{\ln T_{ST} - T_1 - \ln T_{ST} - T_2}{\tau_2 - \tau_1} = \mu_1^2 \frac{a}{h^2},
\]

where \( T_1 \) and \( T_2 \) – temperatures at time \( \tau_1 \) and \( \tau_2 \), \( T_{ST} \) – temperature in the steady state, \( a \) – the coefficient of temperature diffusivity, \( h \) – half-thickness of the sample, \( \mu_1 \) – the first root of the characteristic equation.
\[ \operatorname{ctg} \mu = \frac{\mu}{\text{Bi}}, \]  
(9.2.2)

where \( \text{Bi} = \frac{\alpha h}{\lambda} \) — Biot criteria.

Rate of heating characterizes the rate of change of the excess local temperature \( T \) logarithm, i.e.

\[ -\frac{\partial \ln \left(T_{\text{ST}} - T\right)}{\partial \tau} = m \]  
(9.2.3)

The number \( m \) links the experimentally determined values and the desired thermal characteristics. This relationship exists between the rate of explicit heating (cooling) and the coefficient of thermal diffusivity only at \( \text{Bi} = \infty \). If this condition is not satisfied, the rate of heating, moreover, is implicitly dependent on other properties of the body and the environment, since in this case \( \mu = f(\text{Bi}) \).

Regular thermal regimes have also been in the presence of internal heat sources. They may be uniformly distributed or local.

9.2.1. **Non-stationary methods of determining of thermophysical characteristics in initial stage of heat transfer in the absence of heat sources**

Let us consider technique of definition of the thermal diffusivity on basis of the solution of the differential equation of heat conduction for a semi-infinite body with the boundary condition of the 1st kind.

Body temperature at the initial time is constant and equal to \( T_0 \). At time \( \tau > 0 \) its surface takes the temperature \( T_{\text{ST}} \neq T_0 \), which is maintained constant throughout the experiment.

The general solution of the problem formulated above can be written as:

\[ \theta = \frac{\frac{T}{x, \tau} - T_{\text{ST}}}{T_0 - T_{\text{ST}}} = \operatorname{erf} \left( \frac{1}{2\sqrt{\text{Fo}_x}} \right), \]  
(9.2.3)
where \( \text{Fo}_x = \frac{a \tau}{x^2} \) – the local Fourier number; \( \text{erf} \ z = \frac{2}{\sqrt{\pi}} \int_{0}^{z} e^{-t^2} dt \) – Gaussian error function.

Determination of the coefficient \( a \) is reduced to the recording of the experimental dependence \( \Delta T = T \ x, \tau - T_{st} = f \ \tau \) at a predetermined point \( x \) of the semi-infinite body, the calculation of the relative excess temperature \( \theta = \frac{\Delta T}{T_0 - T_{st}} \) and finding (e.g. table) of the corresponding value of the number \( \text{Fo}_x \). Thermal diffusivity coefficient is calculated by the formula:

\[
a = \frac{x^2}{\tau} \text{Fo}_x , \tag{9.2.4}
\]

where \( x \) – coordinate of the thermocouple junction, recording the temperature difference \( \Delta T \), \( \tau \)– time corresponding to the determined temperature difference \( \Delta T \).

Permanence of excessive temperature at the body surface can be achieved in various ways, for example placing the body between the units via one of which is passed water at constant temperature, different from the ambient temperature in which the sample and a second thermostatted block were initially tested.

9.2.2. **Determination of the thermal diffusivity coefficient in the regular mode**

The differential equation of heat conduction with boundary conditions of the 1st type is in the basis of the method for determining of the thermal diffusivity coefficient.

The plate of thickness \( 2h \) is given. If the plate thickness is small in comparison with length and width then such a plate is generally considered unbounded.

Under the given boundary conditions the heat transfer coefficient is the same for all points of the surface of the plate. The temperature change occurs in only one direction \( x \), in the other two directions temperature does not change \( \left( \frac{\partial T}{\partial y} = \frac{\partial T}{\partial z} = 0 \right) \), consequently, the problem is one-dimensional.

The initial temperature distribution \( T \ x, 0 = T_0 \). At the initial time a constant temperature \( T_{st} = \text{const} \) is defined and maintained on both surfaces.
In the investigation of materials in the regular mode procedure of the experiment is the following. Through blocks connected in series, the water of the constant temperature higher than room temperature on 5-10°C passes. Between the plates and on blocks junctions of the thermocouples are placed, their ends are connected to the ADC and PC. The sample is introduced into the space between the blocks and shrinked by their planes. When testing solid materials it is possible to reduce the thermal resistance by compression of the sample and the preliminary surface treatment of blocks. Moreover, at a sufficiently large flow rates the 1st kind boundary condition is performed with high precision.

Thermal diffusivity coefficient can be found from the relationship:

\[ a = \frac{4h^2}{\pi^2} \frac{\ln \Delta T_1 - \ln \Delta T_2}{\tau_2 - \tau_1} \]  \hspace{1cm} (9.2.5)

The typical dimensions of the sample are known. Changing the absolute excess temperature as a function of time \( \Delta T = f(\tau) \) is determined experimentally. According to the heating curve the graph \( \ln \Delta T = \varphi(\tau) \) is plotted, from which the value of the heat rate is determined:

\[ m = \frac{\ln \Delta T_1 - \ln \Delta T_2}{\tau_2 - \tau_1} \]  \hspace{1cm} (9.2.6)

Thermal diffusivity is calculated by:

\[ a = m \frac{4h^2}{\pi^2} \]  \hspace{1cm} (9.2.7)

To find the dependence \( a = f(T) \) in a wide temperature range multiple repetition of experiments under different temperatures and low temperature variations in the sample are necessary.

9.2.3. **Determination of thermal conductivity coefficient by a comparative method in the steady state**

The most simple execution of the experimental methods are those in which the heat flux passing through the body or bodies of the system remains constant in magnitude and
direction, and a stationary temperature field. The ratio used to calculate the thermal conductivity of the thin plate has the form:

\[ q = \frac{\lambda}{h} (T_1 - T_2). \quad (9.2.8) \]

This equation holds true for the size of specimens for which edge effects are negligible.

According to this equation we can determine the coefficient of thermal conductivity, if we know the value of the heat flow. It is necessary to ensure that the steady state occurred (the temperature at any point of the body or bodies to be constant), and the unaccounted heat loss are small. The latter circumstance requires additional heating elements, which act as protective heaters are designed to protect against heat loss through the side surfaces.

Despite the presence of complex security devices (heaters or a side heat insulation), the determination of the heat flux passing through the sample, is difficult. To avoid them, comparative methods are used. In this methods the material is in contact with the standard. By measuring the temperature difference on the test and reference samples, we can determine the thermal conductivity of the material without measuring the heat flows.

Since in steady-state heat flow through each of the bodies of the system will be the same and constant over time:

\[ \lambda \frac{dT(x)}{dx} = \lambda_s \frac{dT'(x)}{dx}, \quad (9.2.9) \]

Consider a system of two bodies: the standard – the test sample (Fig. 9.2.1).

Fig. 9.2.1. The experimental scheme for determining the thermal conductivity of solid materials
The upper surface of the sample is maintained at a temperature $T_1$, and the lower surface of the standard – at room temperature $T_3$. The densities of heat fluxes passing through the sample and standard are, respectively:

$$q = \frac{\lambda}{h} (T_1 - T_2)$$

(9.2.10)

$$q = \frac{\lambda_s}{h_s} (T_2 - T_3)$$

From the equality of the heat flux densities it follows:

$$\frac{\lambda}{h} (T_1 - T_2) = \frac{\lambda_s}{h_s} (T_2 - T_3)$$

(9.2.11)

whence

$$\lambda = \lambda_s \frac{h}{h_s} \frac{T_2 - T_3}{T_1 - T_2}$$

(9.2.12)

In measuring the thermal conductivity coefficient by a comparative method test material and standard are placed between two blocks throw one of which water is passed from the thermostat with temperature $T_1 = 25-30$ °C, and through second – with room temperature. With the first thermostat temperature higher than room on $5-10$ °C is set. Polymethylmethacrylate (plexiglass) mechanically contacting the test material can be used as a standard. In a steady state temperature difference is measured on the standard and on the test samples. Knowing their thickness and thermal conductivity coefficient of the standard $\lambda_s$ of the test material is calculated by formula (9.2.12).

9.2.4. **Determination of thermophysical characteristics in a medium of constant temperature in the presence of heat source**

Let us consider methods for determining the of thermophysical characteristics under boundary conditions of the 1st and 2nd kind, i.e., in the presence of heat sources and sinks. Under the source the spatially localized heat source of constant electrical power is understood. The simultaneous action of heat sources and sinks leads to a steady-state conditions in which the thermal conductivity coefficient is determined. In all cases, the steady-state is preceded by nonstationary, which is used to determine the thermal
diffusivity coefficient. This creates an opportunity to carry out a comprehensive definition of thermal characteristics in a single experiment.

9.2.5. **Non-stationary methods of determining the thermophysical characteristics in the initial stages of heat exchange**

Let us consider experimental methods in which the sample is considered to be an unlimited body in the direction of the temperature field during the experiment. The advantage of these techniques is to eliminate the boundary conditions of the 1st or the 3rd kind which is difficult to realize with a high degree of accuracy.

The method of constant power source refers to a purely nonstationary methods based on the laws of the initial stage of development of the temperature field in a semi-infinite body heated by a constant power source.

The main advantage of the method is the short duration of the experiment. The disadvantage of this method, as it does all purely nonstationary methods, is the dependence of the final results of the initial thermal state of the sample.

Let us consider semi-infinite body with the initial temperature $T_0$. The bounding surface is heated by a constant heat flux $q = \text{const}$. The temperature change occurs in one direction.

The solution of this problem has the form:

$$
\Delta T_x = T(x, \tau) - T_0 = \frac{2q\sqrt{a\tau}}{\lambda} \text{ierfc} \frac{x}{2\sqrt{a\tau}},
$$

(9.2.13)

where

$$
\text{erf}x = \frac{2}{\sqrt{\pi}} \int_0^x e^{-x^2} \text{d}x,
$$

$$
\text{erfc}x = 1 - \text{erf}x = \frac{2}{\sqrt{\pi}} \int_x^\infty e^{-x^2} \text{d}x,
$$

(9.2.14)

$$
\text{ierfc}x = \int_x^\infty \text{erfc} \xi \text{d} \xi = \frac{1}{\sqrt{\pi}} e^{-x^2} - x\text{erfc}x
$$

If the temperature is measured in the plane of heater ($x = 0$), then (9.2.13) implies
\[
\Delta T_H = T(0, \tau) - T_0 = \frac{2q\sqrt{\alpha \tau}}{\lambda \sqrt{\pi}},
\]  
(9.2.15)

because \( \text{erfc} 0 = \frac{1}{\sqrt{\pi}} \). The relation
\[
\frac{\lambda}{a} = \frac{\lambda c \rho}{b} = b \frac{W \cdot s^{1/2}}{m^2 \cdot K}
\]  
(9.2.16)

characterizes the accumulative thermal ability of the body and is called the coefficient of thermal activity of the body. With the introduction of the coefficient of thermal activity in (9.2.15) we obtain:
\[
\Delta T_H = \frac{2q}{b \sqrt{\pi}} \sqrt{\tau}.
\]  
(9.2.17)

Graphical representation of the equation (9.2.17) in the coordinates \( \sqrt{\tau} = f \Delta T_H \) is a straight line passing through the origin with a slope to the axis of abscissas equal
\[
tg\gamma = \frac{b \sqrt{\pi}}{2q}.
\]  
(9.2.18)

From (9.2.18) we obtain the formula for the calculation of the thermal activity coefficient:
\[
b = \frac{2q}{\sqrt{\pi}} \sqrt{\tau} \Delta T_H
\]  
(9.2.19)

Heat flux \( q \) is calculated by the formula:
\[
q = \frac{I^2 R_H}{2S} = \frac{IU}{2S} \left[ \frac{W}{m^2} \right],
\]  
(9.2.20)

where \( I, U, R_H, S \) – respectively the values of current, voltage, resistance, and area of one side of the heater. The thermal diffusivity coefficient is determined by the delay time, i.e. the time during which the temperature in the section \( x \) becomes the same as the temperature of the sample in the heater plane in the time \( \tau_1 \). For different time points \( \tau_1 \) and \( \tau_2 \) (\( \tau_2 > \tau_1 \)) we can write:
\[
\Delta T_H \tau_1 = \Delta T_x \tau_2
\]  
(9.2.21)

or
\[
\frac{2q\sqrt{a\tau_1}}{\lambda \sqrt{\pi}} = \frac{2q\sqrt{a\tau_2}}{\lambda} \text{ierfc} \frac{x}{2\sqrt{a\tau_2}},
\]  \hspace{1cm} (9.2.22)

therefore

\[
\sqrt{\tau_1} = \sqrt{\tau_2} \cdot \sqrt{\pi} \text{ierfc} \frac{x}{2\sqrt{a\tau_2}}.
\]  \hspace{1cm} (9.2.23)

By making the transformations, we get:

\[
\text{ierfc} \frac{x}{2\sqrt{a\tau_2}} = \frac{1}{\sqrt{\pi}} \sqrt{\frac{\tau_1}{\tau_2}}.
\]  \hspace{1cm} (9.2.24)

We introduce the notation:

\[
\frac{1}{\sqrt{\pi}} \sqrt{\frac{\tau_1}{\tau_2}} = N,
\]  \hspace{1cm} (9.2.25)

\[
\frac{x}{2\sqrt{a\tau_2}} = X.
\]  \hspace{1cm} (9.2.26)

Then from (9.2.24) we obtain:

\[
\text{ierfc}X = N.
\]  \hspace{1cm} (9.2.27)

The value of \(N\) is determined by the formula (9.2.25), which includes the experimentally measured values. Using the table of \(\text{ierfc}X\), the corresponding value of the argument \(X\) is found. This value enables to obtain the estimated ratio for the thermal diffusivity from (9.2.26):

\[
a = \frac{1}{4\tau_2} \left( \frac{x}{X} \right)^2.
\]  \hspace{1cm} (9.2.28)

The coefficient of thermal conductivity and specific heat are found the relations:

\[
\lambda = b\sqrt{a}.
\]  \hspace{1cm} (9.2.29)

\[
c = \frac{\lambda}{ap}.
\]  \hspace{1cm} (9.2.30)

At the same time the calculation of the thermal diffusivity coefficient can be performed on detected experimentally relations of excessive temperatures \(\Delta T_x/\Delta T_H\) at a certain point in time. From the expressions (9.2.13) and (9.2.15):

\[
\frac{\Delta T_x}{\Delta T_H} = \sqrt{\pi} \text{ierfc} \frac{x}{2\sqrt{a\tau}}.
\]  \hspace{1cm} (9.2.31)
We denote the
\[
\frac{1}{\pi} \frac{\Delta T_x}{\Delta T_H} = N_1, \quad (9.2.32)
\]
\[
\frac{x}{2a\tau} = X_1. \quad (9.2.33)
\]

Then
\[
\text{erfc} \; X_1 = N_1. \quad (9.2.34)
\]

Thus, in this case the calculation of the thermal diffusivity is reduced to finding the argument \(X_1\) by the table with the values \(N_1\) found from the experiment corresponding to specific points in time \(\tau\).

In any case, a comprehensive definition of thermal characteristics on the basis of the solution (9.2.13) requires knowledge of the nature of the temperature change over time in any two points of the sample.

**9.2.6. Determination of thermophysical characteristics of solid materials in regular mode**

An unlimited plate with thickness \(2h\) is used as the sample. The initial temperature distribution is uniform and corresponds to the ambient temperature \(T_0 = \text{const}\). At some moment in the plane \(x = 0\) heat source of constant power \(q\) starts to operate. The surface temperatures of the plate throughout the process of heat transfer are kept constant and equal to the initial one. General solution has the form:
\[
T(x, \tau) - T_0 = q \frac{h-x}{\lambda} - 2qh \sum_{n=1}^{\infty} \frac{-1}{\mu_n^2} \cos \left( \mu_n \frac{x}{h} \right) e^{-\mu_n^2 Fo}. \quad (9.2.35)
\]

At \(Fo \to \infty\) (practically at \(Fo \to 2\)) the sum of the series (9.2.35) is negligible compared to the stationary component (first term of (9.2.35)). Therefore
\[
\Delta T_{ST} = T_{ST}(x, \infty) - T_0 = \frac{q}{\lambda} h - x. \quad (9.2.36)
\]

In the regular mode, the thermal diffusivity coefficient is determined from the relation:
\[
a = \frac{4h^2}{\pi^2} \frac{\ln \Delta T_{ST} - \Delta T_1 - \ln \Delta T_{ST} - \Delta T_2}{\tau_2 - \tau_1} = \frac{4h^2}{\pi^2} m. \quad (9.2.37)
\]
where \( \mu_n = 2n - 1 \frac{\pi}{2} \), \( \mu_1 = \frac{\pi}{2} \), 
\( \frac{\ln \Delta T_{ST} - \Delta T_1 - \ln \Delta T_{ST} - \Delta T_2}{\tau_2 - \tau_1} \) the rate of change of temperature for any point in the regular mode, \( \Delta T_{ST} \) – excess steady-state temperature, \( \Delta T_1, \Delta T_2 \) – excess current temperatures.

In studying the materials in the regular mode with the boundary conditions of 1st and 2nd kind, two identical plane-parallel circular plates of equal thickness are used.

Area of the plate should be approximately equal to the area of the flat heater. The heater is placed between the plates. Formed in this way the system is introduced into the space between the blocks and shrinks their planes. Water of constant (room) temperature is passed through blocks from the thermostat.

After leveling the temperature throughout the sample volume (it will be seen from the graphs), the heater is switched off. The choice of the power supplied to the heater is determined by the permissible temperature gradient in the sample, which should be about 10 °C. If the thermal resistance is unknown, even approximately, the desired output is selected by practical consideration.

After switching on the heater excessive temperature starts to rise, reaching a maximum at steady state. Having curve of the temperature difference in time, it is possible to find \( a, \lambda, c \).

In the graphical representation formula

\[
\ln \Delta T_{ST} - \Delta T_i = f(\tau). \tag{9.2.38}
\]

is a straight line.

Therefore, knowing \( \Delta T_{ST} \) and \( \Delta T_i \) (the temperature difference at any point in time) is possible to construct line (9.2.38), the slope of which shows rate of temperature change, and the formula (9.2.37) shows the coefficient of thermal diffusivity.

You can avoid plotting function (9.2.38) and use directly the formula (9.2.37), counting from the time when the current temperature is approximately \( 0,25 \div 0,3 \Delta T_{ST} \), i.e. when it's regular mode.
9.2.7. **Determination of thermal conductivity coefficient in the steady state**

Knowledge of the heat flux through the sample in the stationary mode enables us to find the thermal conductivity by an absolute method, i.e. without using an additional sample with a known thermal conductivity.

Calculated value for calculating the thermal conductivity using a plate as a sample has the form:

\[ \lambda = \frac{q}{T_2 - T_1} \frac{x_1 - x_2}{x_1 > x_2} \]  

(9.2.39)

In order to provide on one surface of the body condition \( q = \text{const} \), we can use the system of the bodies.

Structure of the system is broadly consistent with the formulation of the problem for an infinite plate with a constant source. In the middle of the sample formed by two plane-parallel round or square plates of equal thickness, flat electric heater is placed. To ensure a constant temperature on the grounds of the system, it is placed between two metal blocks and water with constant temperature is passed through the blocks.

If the system is completely symmetrical, i.e. plates have equal thickness and the same properties; the thermal conductivity is calculated by the formula:

\[ \lambda = \frac{qh}{\Delta T_{ST}} = \frac{lUh}{\Delta T_{ST}2S} \]  

(9.2.40)

At different thicknesses but the same properties, the asymmetry of flows can be considered as \( q = q_1 + q_2 \). Moreover,

\[ \frac{q_1}{q_2} = \frac{\Delta T_1 \Delta x_2}{\Delta T_2 \Delta x_1} \]  

(9.2.41)

Only the procedure of measurements is more complicated, since it is necessary to know the temperature changes in the upper and lower areas.

9.2.8. **Pulse methods for determining the thermophysical characteristics**

Patterns of development of nonstationary temperature fields produced by the effect of instantaneous point, line, plane sources of heat in the unlimited body, provided the basis for the generation of pulsed methods for determining the thermal properties of various
materials. The general solution of the differential equation of heat conduction in the case of an instantaneous point source of heat at the point of \((x_1, y_1, z_1)\) of an unlimited body has a view:

\[
\theta = T \cdot x, y, z, \tau - T_0 = \frac{Q}{8\pi \delta \cdot \rho \cdot \pi^2 \cdot \tau - t} \cdot e^{\frac{-x-x_1^2 + y-y_1^2 + z-z_1^2}{4a \cdot \tau - t}}, \quad (9.2.42)
\]

where \(Q\) – the amount of heat generated at the coordinates \(x_1, y_1, z_1\) of an unlimited body at time \(\tau = t\).

The expression for the two-dimensional temperature field \(\theta_1 \cdot x, y, \tau\), generated by the linear instantaneous heat source \(Q_1 \cdot \frac{L}{m}\), operating in moment of time \(\tau = t\) on a line parallel to the axis \(z\) and passing through the point \(x_1, y_1\), obtained by integrating the expression (9.2.42) by \(dz_i\) in the range from \(-\infty\) to \(+\infty\), i.e.

\[
\theta_1 = T \cdot x, y, \tau - T_0 = \left[ -Q_1 \cdot \frac{L}{m} \cdot e^{\frac{-x-x_1^2 + y-y_1^2}{4a \cdot \tau - t}} \cdot dz_1 \right], \quad (9.2.43)
\]

where \(Q_L\) – the amount of heat emitted instantly by unit length of linear heat source at the time \(\tau = t\).

If we integrate the expression (9.2.43) by \(dy_1\) in the range from \(-\infty\) to \(+\infty\), then we get a solution of one-dimensional heat equation for the function \(\theta_2 \cdot x, \tau\) in the case of an instantaneous plane heat source \(Q_S \cdot \frac{J}{m^2}\) in the plane \(x = x_1\):

\[
\theta_2 = T \cdot x, \tau - T_0 = \frac{Q_S}{2\pi \cdot \delta \cdot \rho \cdot \pi^2 \cdot \tau - t} \cdot e^{\frac{-x-x_1^2}{4a \cdot \tau - t}}, \quad (9.2.44)
\]

where \(Q_S\) – the amount of heat generated immediately per unit area of flat heat source at the time \(\tau = t\).

**9.2.9. The method of pulsed linear heat source**

It should be noted that the instantaneous heat source – a mathematical abstraction. In practice, the «instantaneous» is implemented by setting a short heat pulse. Such a pulse may be generated, for example, by the action of an electric or laser source. It is necessary, therefore, to consider the problem of the source action for some time \(\tau_0\) (in formula (9.2.42) \(\tau\) – the time after which the instantaneous source action is observed, but not the
duration of the pulse). For a theoretical solution to this problem, we must make «summation» of instantaneous line source action in time, i.e. integrate (9.2.43) by \( dt \) in the range from 0 to \( \tau_0 \):

\[
\theta = T(x, y, \tau) - T_0 = \frac{q_L}{4\pi\lambda} \int_0^{\tau_0} e^{-\frac{r_0^2}{4a(\tau-t)}} \frac{dt}{\tau-t}, \tag{9.2.45}
\]

where \( r_0 = \sqrt{x^2 + y^2} \), \( x_1 = y_1 = 0 \). Integration of (9.2.45) by the time does not impose any restrictions on the amount of time i.e. on the duration of heat exposure. If the source is valid long enough, we get a curve with a pronounced maximum (Fig. 9.2.2). The temperature at some point will increase monotonically. To obtain the experimental curve similar to Fig. 9.2.2, we must set a short heat pulse. The idea of the method consists in the study of the experimental dependence \( \theta = f(\tau) \) on the extremum, if this relationship is the result of short-term (pulse) heat exposure.

The value of the excess temperature maximum \( \Delta T_{\text{max}} = T_{\tau = \tau_{\text{max}}} - T_0 \) in the point \( r_0 \) will correspond to a specific point in time:

\[
\Delta T_{\text{max}} = \frac{q_L}{4\pi\lambda} \int_0^{\tau_0} e^{-\frac{r_0^2}{4a(\tau-t)}} \frac{dt}{\tau-t}. \tag{9.2.46}
\]

If we differentiate equation (9.2.46) by \( \tau_{\text{max}} \) and equate the derivative to zero, we obtain a relation in which the condition for an extremum must be satisfied:

\[
\frac{d\Delta T}{d\tau} = \frac{q_L}{4\pi\lambda} \tau_0 e^{-\frac{r_0^2}{4a(\tau-t)}} \frac{dt}{\tau-t} = 0. \tag{9.2.47}
\]

Taking the derivative and performing a mathematical transformation, we calculate the ratio of the thermal diffusivity:

\[
a = \frac{r_0^2}{4\tau_{\text{max}}} \ln \left( \frac{\tau_{\text{max}} - \tau_0}{\tau_0} \right). \tag{9.2.48}
\]

Specific heat flow entering the computational relations is calculated by the formula:

\[
q_L = \frac{IU}{L}. \tag{9.2.49}
\]
9.2.10. Method of pulse plane heat source

This method of determining the thermal characteristics based on the solution of one-dimensional heat equation for an unlimited body in the case of action of a flat heat source in it for a short period of time \( \tau_0 \). This solution can be obtained by integrating the expression (9.2.42) by \( dt \) in the range from zero to \( \tau_0 \). If the plane of the source action coincides with the \( yz \) plane and passes through \( x_1 = 0 \), then the value of the maximum of the excess temperature \( DT_{max} = T(x, \tau_{max}) - T_0 \) at a distance \( x \) from the heater will correspond to a specific point in time:

\[
\Delta T_{max} = \frac{q}{2c\rho \pi a} \left[ \frac{x^2}{4a(\tau_{max}-\tau)} \right]^{\frac{1}{2a(\tau_{max}-\tau)}} \frac{d\tau}{\tau_{max}-\tau}. \tag{9.2.50}
\]

If we differentiate equation (9.2.50) by \( \tau_{max} \) and setting the derivative \( \frac{d(\Delta T)}{d\tau} \) to zero, we obtain a condition for an extremum dependence of the excess temperature from the time in point \( x \) of unlimited body, i.e.

\[
\ln \frac{\tau_{max}}{\tau_{max}-\tau_0} = \frac{x^2}{2a} \frac{1}{\tau_{max}-\tau_0} - \frac{1}{\tau_{max}}. \tag{9.2.51}
\]

Equations (9.2.50) and (9.2.51) allow to obtain the following formulas for the thermal characteristics:

\[
a = \frac{x^2}{2 \ln \frac{\tau_{max}-\tau_0}{\tau_{max}}}, \tag{9.2.52}
\]
To measure the thermal diffusivity coefficient by the pulse method, two plates of the test material are placed between the blocks. A thin plate of the same material is placed between them. The source of heat activated for a short time is located at the center of the system.

9.3. **Determination of heat transfer coefficients with the free and forced convection**

**Introduction**

The coefficient of heat transfer $\alpha$ depends on many factors, namely: the rate of fluid flow $\nu$, ambient temperature and a temperature of surface which contacts the environment, its physical properties: viscosity coefficient $\eta$, density $\rho$, heat capacity $C_p$, heat conductivity $\lambda$, and the surface configuration.

Thus, the calculation of the value $\alpha$ presents considerable difficulties; however, the decision of the problem is possible with the help of the theory of similarity.

With regard to physical phenomena concept of similarity applies only to the phenomena of the same kind, described by a set of the same equations.

To derive the equations relating $\alpha$ with different parameters, the function of which is the heat transfer coefficient, the following similarity criteria: Nusselt, Reynolds, Grashof and Prandtl numbers are used.

Reynolds number ($Re$) relates the velocity of the medium and its viscosity and geometrical parameters of the surface in contact with the media:

$$Re = \frac{\nu l}{\eta} = \frac{\nu \rho l}{\eta},$$  \hspace{1cm} (9.3.1)
where \( \eta \) – dynamic viscosity coefficient; \( \nu \) – kinematical viscosity coefficient; \( \rho \) – medium density; \( l \) – the determining geometric size. At the medium movement in a cylindrical tube or at the flow around a cylinder the determining geometric size is the diameter \( d \), and then

\[
\text{Re} = \frac{\nu d}{\nu}.
\]  

(9.3.2)

From the equation defining conditions of surface of a solid heat exchange with a medium Nusselt number (\( Nu \)) is displayed:

\[
\text{Nu} = \frac{\alpha d}{\lambda},
\]  

(9.3.3)

where \( d \) – the determining geometric size (cylinder diameter); \( \alpha \) – coefficient of heat transfer (heat irradiation); \( \lambda \) – coefficient of medium heat conductivity.

For free convection the determining criterion is the Grashof number (Gr):

\[
\text{Gr} = \frac{g \beta \theta l^3}{\nu^2},
\]  

(9.3.4)

where \( \theta \) – the temperature difference between environment and the body surface.

Prandtl number (Pr):

\[
\text{Pr} = \frac{\nu}{a} = \frac{\eta c_p}{\lambda}.
\]  

(9.3.5)

### 9.3.1. Heat irradiation at free convection

The development of free convection of air flow near the heated vertical surface is shown in Fig. 9.3.1. The movement is caused by the Archimedean forces, expelling heated and therefore less dense layers of hot gas near the wall up.
The transition from laminar to turbulent flow is determined by the condition $(Gr \cdot Pr) > 109$.

Fig. 9.3.1 also shows the change in the local heat transfer coefficients along the heated surface.

When regime is laminar heat transfer coefficient decreases in surface height due to increased thickness of the boundary layer. When regime is turbulent $\alpha_x$ practically unchanged with height of the surface, as with the growth of the boundary layer thickness the intensity of the turbulent transport increases.

When regime is laminar thickness of hydrodynamic $\delta$ and thermal $\delta_t$ boundary layers are almost equal. The distribution of temperature and velocity within the boundary layer has the form shown in the graph in Fig. 9.3.2. The maximum flow rate falls on the cross section located at a distance of $0.38\delta$ from the surface.
Under the assumption of a uniform motion layer under the action of counterbalancing the Archimedean and viscous forces on a wall the similarity equation

$$\text{Nu} = f(\text{Gr} \cdot \text{Pr})$$  \hspace{1cm} (9.3.6)

is true.

Local heat transfer coefficient on the vertical surface may be calculated by the following empirical formulas:

- laminar regime \(((\text{GrPr})_x \leq 10^9, q = \text{const})$$:
  $$\text{Nu}_x = \frac{\alpha_x}{\lambda} = 0.60(\text{Gr} \cdot \text{Pr})_x^{1/4} \varepsilon_T; \hspace{1cm} (9.3.7)$$

- turbulent regime \(((\text{Gr} \cdot \text{Pr})_x > 10^9$$):
  $$\text{Nu}_x = \frac{\alpha_x}{\lambda} = 0.15(\text{Gr} \cdot \text{Pr})_x^{1/3} \varepsilon_T; \hspace{1cm} (9.3.8)$$

where \(\varepsilon_T = (\text{Pr}_M/\text{Pr}_W)^{1/4}\) – correction for changing the physical parameters depending on the temperature; for gases \(\varepsilon_T = 1\).

The temperature far from the wall is the determining temperature at which the values of the physical parameters of the coolant fluid are selected. The only exception is the value of \(\text{PrW}\) – index \(\text{W}\) indicates that in this case the heat transfer medium Prandtl number must be calculated at the wall temperature. The value of heat transfer coefficients for laminar regime depends on the boundary conditions on a wall: at a constant temperature of the surface \(\alpha x\) value approximately 14\% lower than at a constant heat flux density at the wall. For the turbulent regime, this dependence is practically not observed.

Free convection around a horizontal pipe usually occurs in the laminar regime, since the surface has a small length range adjustment. Average heat transfer coefficient calculated by the formula:

$$\text{Nu}_d = \frac{\alpha d}{\lambda} = 0.50(\text{Gr} \times \text{Pr})_d^{1/4} \varepsilon_T . \hspace{1cm} (9.3.9)$$

The relations given above are valid for gases and non-metallic liquids.

9.3.2. **Heat irradiation at forced convection**

Fig. 9.3.3 illustrates how the boundary layer develops and a heat transfer coefficient varies along the plate.
The transition from laminar to turbulent flow is determined by the value of the Reynolds \( \text{Re}_{x} \approx 3 \times 10^{5} \).

When flow is laminar (\( \text{Re}_{x} < \text{Re}_{x_{\text{cr}}} \)) and the boundary condition \( T_{ST} = \text{const} \) local heat transfer coefficient is calculated by the formula

\[
\text{Nu}_{x} = \alpha_{x} \nu / \lambda = 0,332 \ \text{Re}_{x}^{0.5} \ \text{Pr}^{0.33} \ \epsilon_{T}.
\] (9.3.10)

The condition \( q = \text{const} \) (electric heating of surface) is often implemented in the experiments. Using the law of heat exchange and the integral equation of the thermal boundary layer by \( q_{ST} = \text{const} \) we obtain:

\[
\text{Nu}_{x} = 0,469 \ \text{Re}_{x}^{0.5} \ \text{Pr}^{0.33}.
\] (9.3.11)

As it can be seen from a comparison of the last two equations, the local heat transfer coefficients in laminar regime strongly depend on the type of boundary conditions (the difference is about 40%).

For a turbulent boundary layer (\( \text{Re}_{x} > \text{Re}_{x_{\text{cr}}} \)), emerging from the front edge of the plate, the following design relations are true:

\[
\text{Nu}_{x} = 0,0296 \ \text{Re}_{x}^{0.8} \ \text{Pr}^{0.4} \ \epsilon_{T} \ (T_{C} = \text{const}),
\] (9.3.12)

\[
\text{Nu}_{x} = 0,031 \ \text{Re}_{x}^{0.8} \ \text{Pr}^{0.4} \ (q_{C} = \text{const}).
\] (9.3.13)
If there is a section on the plate laminar flow of a large extent, it is necessary to consistently calculate the heat transfer in the laminar and turbulent regions and use the conjugation condition.

9.3.3. **Heat irradiation at forced transverse flow around a cylinder**

A characteristic feature of the flow around a cylinder is the separation of the boundary layer to form a vortex flow in the feed zone. At \( \text{Re}_d < \text{Re}_{d_{cr}} = 2 \cdot 10^5 \) (subcritical flow) in the front of the cylinder formed by the laminar boundary layer separation, followed by \( \phi = 80^\circ - 90^\circ \) (Figure 9.3.4a). In a supercritical flow \( \text{Re}_d > \text{Re}_{d_{cr}} \) for some \( \phi \leq 90^\circ \) there is a transition from laminar to turbulent flow in the boundary layer the separation point moves downstream \( (\phi_{SEP} = 140^\circ) \), Fig. 9.3.4b. The phenomenon of separation is explained by the «reverse» pressure drop in the feed zone of the cylinder, under the influence of which inhibited by the viscous forces near-wall layers of the medium is turned back. In the turbulent boundary layer momentum exchange between the external thread and wall layers is so intense that remains unseparated for even in the inverse differential pressure.

![Fig. 9.3.4. Subcritical (a) and supercritical (b) flow around a cylinder](image)

The above-mentioned features of hydrodynamics determine the nature of changes in the heat transfer coefficient around the circumference of the cylinder.

Reducing the heat transfer coefficient in the frontal part is explained thickening of the laminar boundary layer. The appearance of the maximum heat transfer coefficient associated with the transition from laminar to turbulent flow in the boundary layer.

To calculate the cylinder coefficient of heat transfer average value at low free-stream turbulence, the following formula (which determines is the average temperature of the medium) are used:
\[ \text{Nu}_d = 0.5 \text{Re}_d^{0.5} \text{Pr}^{0.38} \varepsilon_T \text{ (when } 5 < \text{Re} \leq 10^3), \quad (9.3.14) \]

\[ \text{Nu}_d = 0.25 \text{Re}_d^{0.6} \text{Pr}^{0.38} \varepsilon_T \text{ (when } 10^3 < \text{Re} \leq 2 \cdot 10^5), \quad (9.3.15) \]

\[ \text{Nu}_d = 0.023 \text{Re}_d^{0.8} \text{Pr}^{0.37} \varepsilon_T \text{ (when } 3 \cdot 10^5 \leq \text{Re} \leq 2 \cdot 10^6). \quad (9.3.16) \]

### 9.4. Methods for determination the heat capacity

**Introduction**

In the process of the phase transitions studying, drawing heat balance processes, in the design of industrial and civil construction, inspection and testing of product quality the heat capacity of materials is an important parameter.

The heat capacity of a substance $C$ is one of the most important thermodynamic physical parameters the value of which is used for determining the entropy, enthalpy, Gibbs energy, and other variables.

The heat capacity of the body can change its meaning in different contexts, which finds wide application: in the studying of the substances structure and their properties, in the studying of phase transitions and critical phenomena, calculating the total amount of impurities in the material, determining the thermal effects of chemical reactions. The heat capacity is also an important characteristic in calculating of heat loss walling and preparing balance sheets of thermal units.

The ratio of $C$ is characterized by the ratio of the quantity of heat $Q$, transferred to the body (system) in any process, to the corresponding change in temperature $dT$.

Heat capacity is a measure of the energy required to raise the temperature of the material. This energy is used to increase the energy of the vibrational motion of the atoms about their equilibrium positions in the lattice, increasing the energy state of some of the electrons in the lattice, changing the positions of the atoms (in the formation of structural defects or changes in the structure).

The ratio of heat capacity to body mass ($m$) are is called specific heat capacity ($c$), and the ratio of heat to the amount of substance ($\mu$) in moles is called molar heat capacity ($c_m$).

The heat capacity depends not only on the initial and final states of the body, but also the manner in which the transition was made between them.
Usually distinguish the heat capacity at constant pressure $C_p$ (isobaric process) and at constant volume $C_V$ (isochoric process).

The difference between the two processes is in the fact that heating in the first case ($p = \text{const}$) portion of the heat goes to the production of work to expand the body and another portion of the heat goes to increase the internal energy; whereas in the second heating ($V = \text{const}$) all the heat spent to increase the internal energy of the body.

The difference between these values is low in solids at low temperatures, but at high temperatures, it may be significant. For ideal gases, $C_p - C_V = R$ – universal gas constant equal to 8.314 J/mol·K.

Heat capacity depends not only on how body was heated, but on the macrostructure, chemical composition, and physical state of the body.

The dependence of the heat capacity on temperature is quite complex. It is explained by the basic concepts of quantum theory and characterized by «Debye temperature». At the same time heat capacity is proportional to temperature only at low temperatures. Heat capacity does not depend on the structure of the crystal lattice, but increases at its destruction.

Organic materials have significantly higher specific heat capacity than the mineral. It’s possible to submit the following conditional number of construction materials with different chemical compositions on the specific heat capacity (kJ/kg·K at $t = 25 \, ^\circ\text{C}$):

- iron .................................................. 0,45
- steel .................................................. 0,48
- granite.............................................. 0,65
- concrete, cement, lime, gypsum........... 0,84
- glass, construction ceramics.............. 0,88
- limestone .......................................... 0,92
- typePVCfoams.................................... 1,26
- styrofoam.......................................... 1,34
- polyurethane foam .............................. 1,47
- bitumen, phenolics............................ 1,68
- wood, wood fiber............................... 2,30
Due to the chemical nature some materials are capable to transmitt energy, remaining stable, and others – to accumulate it until their destruction. Inorganic materials, atomic structure which has a wave nature, are conductors of heat, and organic matter – accumulators or heat insulators.

By this criterion the specific heat \((c)\) has a relationship with thermal conductivity \((\lambda)\) and temperature conductivity \((a)\).

If to compare the specific heats of different substances with identical chemical compounds in different aggregate states, their values are very close. The main factor is the chemical composition. Some results of the comparative evaluation:

- Gases (except for rare), such as air, oxygen, hydrogen and nitrogen having a specific heat capacity \(c \approx 0,92 \text{kJ/kg·K}\), i.e. like limestone;
- Liquids from benzene \((c = 1,35 \text{kJ/kg·K} – \text{minimum value})\) to ethanol \((c = 2,42 \text{kJ/kg·K} – \text{maximum})\) have approximately the same specific heat capacity such as organic polymeric materials of foam \((c = 1,26 \text{kJ/kg·K})\) to wood \((c = 2,30 \text{kJ/kg·K})\). Even the extreme values of \(c\) for the liquid (mercury) and solid (lead) equal and amount to \(0,13 \text{kJ/kg·K}\).

It’s necessary to note the abnormally high specific heat capacity of water \((c = 4.18 \text{kJ/kg·K})\), that should be taken into account in the design and calculation of thermal plants for drying and thermal-processing of building materials. Humidification material leads to a considerable increase of their specific capacity and, consequently, increases the consumption of energy by heat treatment.

To determine the heat capacity of substances the special equipment – calorimeters are used.

Theoretical basis of calorimetry based on the heat balance equation, which can be written as:

\[
Q = Q_1 + Q_2 + Q_3, \tag{9.4.1}
\]

where \(Q\) – the total heat evolved in the system; \(Q_1\) – the amount of heat accumulated by calorimetric vessel material; \(Q_2\) – the amount of heat accumulated by the sample; \(Q_3\) – heat losses to the environment; and on a known ratio:

\[
Q = cm \ T_1 - T_2, \tag{9.4.2}
\]
whereby, when the body is heated from an initial temperature $T_1$ to the final temperature $T_2$ this body accumulates the amount of heat $Q$, where $c$ – specific heat capacity; $m$ – mass of the body.

The calorimeter construction is determined by the temperature range and the desired accuracy. Modern calorimeter operating in the temperature range from 0.1 to 3500 K. There are (boundaries are conditional) low temperature ($\leq 90$ K), middle range ($90$ K $\leq T \leq 500$ K) and high temperature calorimeters ($T \geq 500$ K).

The calorimeter includes a calorimetric vessel in which the process takes place under study, the temperature sensor (mercury thermometer, resistance thermometer, thermocouple or thermopile, thermistor, quartz thermometers and the others; at temperatures above 1300 K optical pyrometers are used), electric heater, etc. Calorimetric system is protected by the screens or enclosures designed to regulate its heat exchange with the environment. Shells can be adiabatic or isothermal. The temperature difference between the calorimeter system and shell is controlled thermocouples, thermistors, etc. The temperature of the shell provided with an electric heater is controlled automatically by the electronic devices.

In liquid calorimeter the vessel is filled by a certain amount of so-called calorimeter liquid (usually distilled water, at least – ethanol, liquid nitrogen, liquid paraffin, molten tin, etc.). A calorimeter bomb or ampoule with the substance is placed into the vessel. These calorimeters are most often used to work at room temperature to measure the heat capacity of solids and liquids.

To determine heat capacity of solids and liquids in the range of 0.1 to 1000 K calorimeters-containers are used. The calorimetric vessel is the thin-walled container (vial for material) in this calorimeters, it is typically small (0.3 to 150 cm) made of copper, silver, gold, platinum, stainless steel.

Calorimeters-containers, designed to operate at lower temperatures besides the isothermal or adiabatic shells systems are protected by the vacuum jacket and placed in a cryostat (Dewar flask) filled depending on the temperature of liquid by helium, hydrogen or nitrogen. For operating at elevated temperatures it is placed in a thermostated electric furnace. The heat capacity is determined by periodic or continuous feed of heat method.

The most accurate measurements of heat capacity of substances are generally carried out in an adiabatic calorimeter (Fig. 9.4.1).

The sample 1 of the test material (liquid or solid) with a known mass $m$ and the specific heat $c_x$, to be measured, is placed in a glass ampoule2 with known specific heat capacity $c_a$
and mass \( m_a \). The heater 3 can be placed in the wall of the ampoule 2 (if the sample 1 is a solid) or directly in the material, if it is liquid. Power meter 4 measures the electrical output \( W(\tau) \), consumed by the heater 3, and calculates the total amount of heat:

\[
Q = W \tau \, d\tau
\] (9.4.3)

let down to a bipartite core formed by the sample vial 1 and 2. Thermocouple 5 and 6 secondary device can measure the temperature of the bipartite core. Fig. 9.4.1 shows that the thermocouple 5 may be mounted directly in the material (if liquid or granular material), however, when measuring the heat capacity of solids, thermocouple 5 is often installed in the wall of the ampoule 2.

![Diagram](image)

Fig. 9.4.1. The model of adiabatic calorimeter: 1 – the sample of unknown specific heat capacity \( c_x \), and a known mass \( m \); 2 – ampoule (glass cover); 3 – heater; 4 – a device for measuring electric power (power meter); 5, 8, 9 – thermocouples; 6 – a device for measuring the temperature; 7 – outer metal shell; 10 – automatic controller that supports zero temperature difference.

If during the experiment a perfect thermal contact between the test sample 1 and ampoule 2 provided, the temperature of the sample and the tubes can be considered the same. In the total absence of heat loss to the environment as heat balance equation for the bipartite core can be written as:

\[
W \tau \, d\tau = Q = c_x m \, T_B - T_E + c_a m_a (T_B - T_E),
\] (9.4.4)

where \( Q \) – the total amount of heat supplied to the heater 3; \( c_x, m \) – specific heat capacity and mass of the sample 1; \( c_a, m_a \) – specific heat and mass of the ampoule 2 with built-in heater and cover; \( T_B, T_E \) – temperatures recorded by the thermocouple 5 at the beginning and the end of the experiment.

The outer shell 7 is typically made from metal, which allows one to create isothermal conditions around the central core of the bipartite. Adiabatic system, which includes a differential switched on thermocouples 8, 9, an automatic regulator 10 and protective
heater 11, provides an almost complete elimination of heat loss from the bipartite core (including the test material 1 and glass ampoule 2 with cap) by maintaining a zero difference temperature between the outer metal shell 7 and the ampoule 2.

The full ampoule 2 heat capacity (together with the heater 3 and the cover) is revealed into practice in the preliminary experiments and designated:

\[ C_a = c_a m_a. \]  

(9.4.5)

Then the heat balance equation is written as

\[ Q = c_x m \ T_1 - T_2 + C_a (T_1 - T_2). \]  

(9.4.6)

From equation (9.4.6) is obtained

\[ Q = T_1 - T_2 \ c_x m + C_a , \]  

(9.4.7)

whence the basic formula for the calculation of the adiabatic calorimetry is followed:

\[ c_x = \frac{1}{m} \ \frac{Q}{T_1 - T_2} - C_a. \]  

(9.4.8)

In most instances, we cannot completely eliminate the leakage of heat from the bipartite central core to the environment. Then the heat balance equation takes the form:

\[ Q = c_x m \ T_1 - T_2 + C_a \ T_1 - T_2 + \frac{\tau}{\tau_0} k \ T_c \ \tau - T_E \ \tau \ d\tau, \]  

(9.4.9)

where \( k \) – coefficient of effective thermal conductivity of the environment; \( T_1 \) – the central core initial temperature; \( T_c \ \tau \) – core (ampoule) temperature change with time; \( T_E \) – environmental temperature. The third term in the last equation represents the heat losses to the environment.

The glass ampoule 2 with an efficient heat capacity built in heater 3 is expedient to determine in experiments without the sample.

Adiabatic calorimeters are quite simple design; so they can measure the heat capacity with a small margin of error, making them suitable for high-precision scientific research.

When measuring the specific heat the double calorimeter is also used; it has two identical calorimeter system (liquid, solid, thin), which are at the same temperature and have the same heat exchange with the shell. Instead of the amendment to the heat exchange a small correction to the non-identity of the calorimetric systems (units) determined in advance is introduced. In constant temperature or isothermal calorimeters heat quantity is measured by
the amount of the substance has changed its physical state (melting ice, naphthalene or evaporation of the liquid.

To determine the heat capacity of substances with significant conductivity (metal alloys) at temperatures up to 4000 K, modulation and impulse calorimetry methods are used. In the first method the sample temperature oscillation amplitude is measured when passing AC of known frequency through it; in the second one – the temperature rise during heating of a thin wire (or rod) made from a sample by current pulses. The method of calorimetry with flash laser heating which is used for the study of metal and ceramic materials as well as liquids in the temperature range 80 – 1100 K refers to impulse calorimetry.

In contrast to the quasi-static calorimetric methods dynamic methods for the determination heat capacity are also used. Their underlying equation relating the required heat capacity with experimentally measured parameters, obtained from the solution of unsteady heat conduction equation. These new methods have two fundamental common features:

- Methods of measuring heat capacity based on analytical solution of the heat equation with different boundary and initial conditions;
- Techniques anyway based on the forced transfer of the sample (by a predetermined amount of heat absorption) from the initial state with the temperature $T$ in the state close to it with the temperature $T + \Delta T$.

The diversity of the developed methods is to measure the specific heat capacity due solely by differences in the types and methods of heat stress on the sample, the heat capacity of which is the required value.

Dynamic methods used to determine the heat capacity of substances, depending on the thermal effect on the sample, can be divided into three groups:

- Modulation method;
- Pulse method;
- Method of monotonic warming.

The modulation method consists in creating periodic oscillation supplied to the sample and recording power occurring with variations in the sample temperature. Unilateral supply of power to the sample is reborn as a method of temperature waves. This method is one of the most informative, as it allows to measure factors other than the heat capacity, thermal
and temperature conductivity, thermal expansion coefficient and a number of others. Method used in a wide temperature range: from tenths Kelvin to the melting point of refractory metals. This method is most effective for the study of metals and alloys, i.e. the electrical conductors when direct heating of electric current sample is used. For samples being insulators separate heaters are used. In this case, the sample is not affected by the volume harmonic power source but by the surface. In the peripheral location relative to the heater layers of the sample temperature damped waves are propagating. There are variations of modulation calorimetry when determining the amplitude of oscillation is replaced by measuring the temperature of the sample frequency and the phase shift. Achieved in the moment measurement accuracy by such methods is not too high: 3 – 4% for metals, 5 – 7% for dielectrics. However, with appropriate elaboration of metrology, the advances of modern technology and electronics can be expected to achieve an accuracy of 0.5% and above.

Pulse method is based on the rapid heating of the sample under the influence of the heat pulse and analysis of temperature change over time. Pulse methods are especially useful for the study the heat capacity and other thermal properties under extreme conditions (near the melting point, in terms of heat stroke, etc.), when the prolonged maintenance of the system at such a high temperature level is technically very difficult task. Currently, this method is developed for metals and alloys, and gives an error of 1 – 2%. When using pulsed methods, it should be kept in mind that they are often not applicable to materials with long relaxation time of thermal processes, such as polymers because of the abrupt sample temperature changes over time.

The method of monotonic warming is based on the laws of regular thermal regime in which the temperature field of the sample ceases to depend on the initial conditions. The heat capacity measurement is based on the ratio:

$$C_a = \frac{P(\tau)}{dT/d\tau},$$

(9.4.10)

where $P(\tau)$ – power applied to the sample, $dT/d\tau$ – heating rate.

The monotone heat method is implemented in heat capacity meter IT-s-400, designed for the heat capacity temperature dependence of solids, dry, fibrous materials, liquids with a density of 800 kg/m$^3$ investigations.

In the process of preparing for work the sample is placed in an ampoule, made in the form of a cup with a lid. A glass is placed on a heat meter fixed to the ground with a massive built-in heater. Thermal coupling of the sample (located in a glass with a lid) with
the external environment is only allowed through the heat meter. Therefore, the outer surfaces of glass separated from the environment adiabatic shell formed as a cap with a built-in heater. The automatic control system maintains a zero temperature difference between the cap and the glass by changing the electrical power supplied to the heater, which is evenly distributed over the lateral surface of the cover. Monotonic heating of the sample and the tubes is carried out by supplying electric power to the heater that is built into the base.

The quantity of heat that passes per unit time through a heat meter section is consumed both on heating of the sample, and the heating of a glass ampoule with a cover:

$$P_T = P_0 + P_a,$$  \hspace{1cm} (9.4.11)

where \(P_0\) – power (amount of heat per unit time), going on heating the sample; \(P_a\) – power (amount of heat per unit time), going on heating the ampoule; \(P_T\) – full power (amount of heat per unit time), passing through the heat meter.

The values of \(P_0, P_a\) can be calculated according to the formulas:

$$P_0 = cm_0b, P_a = c_ah_a b,$$  \hspace{1cm} (9.4.12)

where \(c, c_a\) – specific heat capacity of the sample and the ampoule \([\text{J/kg} \cdot \text{K}]\); \(m_0, m_a\) – mass of the sample and the ampoule \([\text{kg}]\); \(b = \frac{dT}{dt}\) – heating rate \([\text{K/s}]\); \(C_a = c_am_a\) – ampoule total heat capacity \([\text{J/K}]\).

Total heat capacity \(C_a\) is a constant characteristic of the ampoule designed as a cup with a lid, and depends only on the temperature \(T\).

The magnitude of the thermal power \(P_T\), passing through the heat meter, can be determined from the change in temperature on heat meters \(\theta\) and from the heat meter thermal conductivity coefficient \(k_T\) identified in the independent calibration experiments.

Parameter \(k_T = k_T(T)\) is a constant characteristic of a heat meter and depends only on the temperature \(T\). Thus, the heat balance equation for the ampoule with the sample placed inside it can be written as:

$$k_T\theta = cm_0b + C_a b,$$  \hspace{1cm} (9.4.13)

whence we get the calculating formula:

$$c = \frac{1}{m_0} \frac{k_T\theta}{b} - C_a.$$  \hspace{1cm} (9.4.14)
The greatest development of the monotonic warming method has received in a variant of a differential calorimeter, where nonadiabaticity could be neglected.

There are the so-called differential scanning calorimeters. In these devices in the process of monotonic warming the power difference needed to heat the two samples at a given speed is measured. Thus, the heat capacity of one of the samples is known. At present, devices based on this method have been used in various fields of science and technology, including industrial production for quality control of products. Factors influencing the wide dissemination of these instruments were as follows:

- Resolution, or the response threshold, located at the level of units of microwatts;
- High performance;
- Small (up to several milligrams) mass of samples, which is especially important in studies of rare or expensive to obtain materials;

Highly informative, allowing in one experiment to determine the number of parameters (specific heat, physicochemical process heat, the kinetic parameters, the degree of the material purity).
10. Simulation of fluid dynamics, heat and mass transfer in flows

Andrei Chorny
Belarusian State University, Physical Faculty
Introduction

For a better understanding of technological processes and for a more successful designing of new engineering facilities we need a more detailed description of fluid flow properties with regard not only to averaged velocity, temperature or concentration values but also to fluctuation characteristics of considered parameters. Problems of organizing the operation of different heat and mass exchange apparatus are the examples of serious technical gaps and of contradictions that impose severe demands on fluid dynamics theory. Demands to enhance more complete fuel combustion, to reduce the pollutant content, and also to provide an alternative use of fuel resources create a need for optimization of combustion conditions in operating volumes and channels of engine combustion chambers, in industrial furnaces, power plants, chemical reactors, etc. that in turn specify their energy and ecological properties.

The problems mentioned above are being solved by means of obtaining and collecting detailed data on physical phenomena. In doing so, the first estimation of the processes of fluid dynamics, heat and mass transfer is usually made by numerical simulation.

Fluid dynamics, heat and mass transfer studies are based on extensive experimental experience and theoretical approaches invoking a broad spectrum of mathematical tools – from simple algebraic relations to probability methods. Within the modern state-of-art of this area of knowledge, the methods and notions of classical fluid dynamics are combined with the ideas of statistical fluid mechanics and computational mathematics (Лапин, 1989; Libby, 1994).

A mathematical description of fluid flows starts from the equations of continuity, Navier – Stokes, energy and mixture species transfer, as well as from thermodynamic and kinematic relations. Solving such a system subject to appropriate boundary conditions yields total information about space-time distributions of velocity, pressure, temperature, density, and chemical composition. However practically, it is difficult to do because of a wide range of flow length and time scales, especially for turbulent flows. A chemical reaction determines scales associated with the kinetics of elementary reactions (their space scale is usually less than the Kolmogorov length scale). This circumstance imposes additional restrictions on the resolution of a computational grid. To date, having developed in relation to computer technology achievements, direct numerical simulation (Moin, 2004) of complex flows cannot be used in full measure first for engineering problems (Протодьяконов, 1983). The methods mentioned here find use mainly with research objectives for simple flow geometry configurations at moderate Reynolds numbers (Re). That is why, statistical methods (Large Eddy Simulation (Ferziger, 1987) and classical
Reynolds approach (Лапин, 1989; Libby, 1994)) are under development, including three main aspects of modeling: chemical kinetics, mutual influence of fluid dynamics and chemical processes, and also directly of turbulence (Fox, 2003).

A necessity to model chemical kinetics is caused by the scarcity of CPU time. An account of a detailed mechanism of reactions is justified for chemical systems with a small number of reactive components. In other words, one should consider assumptions which will allow simplifying the description of such systems: the assumption of local chemical equilibrium should be introduced for fast reactions; the state of a reactive medium should be characterized involving special parameters – mixture fraction, Shvab–Zeldovich variables, progress variable, etc. (Libby, 1994).

Recently, the mathematical apparatus of fluid dynamics theory has been developed to a level, at which it is in principle possible to pose and solve practically important problems with the use of fluid dynamics and thermodynamics equations (Лапин, 1989; Libby, 1994; Fox, 2003). In this situation, usually the dynamic parameters (velocity, pressure, density, temperature, concentration, etc.) are regarded as random space and time functions. Depending on the parameters chosen, different approaches to the study of fluid dynamics (Коловандин, 1980) are known and are cited below since the completeness of the statistical description of random fields reduces to:

- functional approach using a probability functional of a random field (Монин, 1967);
- approach based on a probability density function (PDF) (Libby, 1994; Fox, 2003; Pope, 1985; Dopazo, 1994);
- spectral approach considering energy spectra in the space of wave numbers or frequencies (Монин, 1967; Фрост, 1980);
- correlation approach dealing with the simplest characteristics of a random field – one-point correlation functions and statistical moments (Лапин, 1989; Libby, 1994; Fox, 2003; Коловандин, 1980).

Over the last three decades, the PDF method of fluid-and thermodynamic quantities (Libby, 1994; Pope, 1985; Dopazo, 1994) has been actively developed. It is identical to the method of kinetic equations in statistical physics (Гардинер, 1986), e.g., in theory of rarefied gases. Realization of a large part of problems by this method is based on postulating a PDF form or on building up closed transport equations for the PDF in the composition space of probability variables and on their solving by computational mathematics approaches.
When the turbulent flow of a continuous medium with chemical reaction proceeding in it is under study, the equations for temperature and concentration PDFs of mixture species are equivalent instead of those for heat and mass transfer in the laminar flow (in combustion studies – theory of laminar flames). In such a situation, the mathematical models for different turbulent reactive flows, which contain the systems of equations for statistical moments, are applied in combination with the probability approaches, in particular with the PDF method (Libby, 1994; Fox, 2003; Veynante, 2002). The advantage of the latter consists in the faithful representation of the influence of chemical source terms on unknown quantities. Here, calculation of correlations, which allow determining an averaged chemical reaction rate, requires a more thorough description of the process of fine-grained mixing (micromixing) of a scalar field of reactant concentration and temperature characterized by the small-scale turbulent transfer and molecular diffusion interaction (Колмогоров, 1941; Хинце, 1963). Indeed, for chemical reaction to proceed, flow reactants must be in the stoichiometric ratio. Such an event may take place only after the strain and extension of a surface contacting different-concentration regions through a fluctuating velocity field have been followed by increase in its area, and this provides favorable conditions for molecular diffusion manifestation. It assures a uniform concentration distribution at molecular scales, whereon the chemical reaction is possible. The chemical reaction rate is thus presumed not so much by kinetic parameters (energy activation, thermal effect of a reaction, etc.), as by mixing conditions of mixture components, i.e., by physical parameters, in particular by fluid dynamics.

So, we focused our attention on more complex problems of fluid dynamics – the description of turbulent transfer involving various additional physical processes (on example, chemical reactions) and also on the related closure problem faced by statistical approaches. These approaches to fluid flows have been applied along with classical models – Reynolds-Averaged Navier–Stokes (RANS) approach. The last decades have been marked by the use of Large Eddy Simulation (LES) to fluid dynamics, heat and mass transfer, combustion.

As for the velocity field, in the 1960s the LES technique was adopted as a new modeling approach (Smagorinsky, 1963). Unlike statistical approaches in terms of Reynolds stresses, the LES can give large-scale unsteady motions in explicit form. In particular, this is beneficial for reactive jets, bluff-body flames, and swirling flames in which such motions are evident. Furthermore, the unsteady LES is best suited for the study of non-premixed combustion dynamics (Stone, 2002; Varoquie, 2002).
As for the interaction between chemical reactions and turbulent flow, the LES does not seem to be a new approach. This follows from the fact that these two processes interact, predominantly, at the smallest, unresolved scales. So, we can say that the LES faces the same closure problem as the RANS-based approach and needs the use of similar Sub-Grid-Scale (SGS) models, e.g., SGS flamelet models (Pitsch, 2000; Kempf, 2002). The SGS model demands can be small since the amplitudes of subgrid fluctuations are less than those of the Reynolds-averaged ones. Also, the resolved fields provide additional information that can be incorporated into the SGS modelling (Pitsch, 2002). Because of substantial computational LES requirements, quite simple turbulent-combustion SGS models are widespread. But at the present moment, the LES technique combined with more comprehensive SGS models (e.g., CMC (Bushe, 1999), PDF (Sheikhi, 2005), and the linear eddy model (LEM) (Menon, 1996; Chakravarthy, 2000)) finds a wider use. LES computations have been performed of practical combustors (Kim, 1999; Moin, 2004; di Mare, 2004; Selle, 2004) and the obtained results have been checked as against flow field and temperature data. However, the quantitative testing of the SGS models for fluid dynamics and combustion in comparison with detailed experimental data is insufficient.

10.1. Fluid flow features and parameters

10.1.1. Basic turbulence contribution to fluid flow processes

Turbulent motion is one of the major movements of a continuous medium frequently encountered in nature and technology. Unlike the other major movements – laminar motion – characteristic for turbulence random changes in medium properties enhances much mixing processes of liquid or gas (Brodkey, 1975 (a); Baldyga, 1999). Mixing is an important phenomenon occurring in the turbulent flow. It falls into a class of physical phenomena to be characterized by a close interaction of the turbulent motion with chemical and thermodynamic processes in the flow field (Brodkey, 1975 (a); Baldyga, 1999; Brodkey, 1975 (b)). When such efficient mixing is absent, many processes associated with the flame onset and chemical reactions, as well as with admixture spread (Лапин, 1989; Libby, 1994; Ferziger, 1987) are impossible in the environment and in engineering facilities. Owing to available turbulent diffusion, the turbulent motion augments heat/mass transfer and permits essential variations in the chemical interaction rate of species to be transported. As a result, turbulent mixing has become an integral part of high-efficient chemical technologies, which places strong applied and fundamental emphasis.
10.1.2. Distinctive features of laminar and turbulent transfer

In applied problems of fluid dynamics, heat and mass transfer, turbulent mixing is interpreted as the disappearance of inhomogeneities not only of a flow velocity $U_i$ and a pressure $p$, but also of a definite set of scalar parameters: density $\rho$, temperature $T$, concentration $Y_\alpha$ of species to be transported, etc. Heat and mass transfer is governed by varying a scalar field. By a scalar is understood either a physical parameter (temperature or concentration) or a specially constructed variable (mixture fraction $f$, Shvab–Zeldovich variables $\beta_\alpha$, progress variable $Z$, etc.) (Libby, 1994; Fox, 2003).

In chemically reactive systems (Libby, 1994; Fox, 2003; Veynante, 2002), the mixing process is also treated more narrowly just as the change in the inhomogeneous state of composition reactants into the homogeneous state, i.e., the formation of homogeneous concentration in the scalar field when the chemical reaction is most likely. Since temperature and concentration variations of reactants and reaction products are fluctuating in character, the transport of heat and mixture species has a complex dynamic behavior closely related to random velocity fluctuations. Scalar turbulence is said to exist (Shraiman, 2000). That is why, to model turbulent flow and processes in scalar fields, it is first necessary to understand and forecast mechanisms of the turbulence influence on their mixing.

In laminar flow, scalars are propagated due to mechanical transfer through the main flow, due to strain under the action of velocity gradients, as well as due to molecular diffusion and heat conduction that define a final location of liquid or gas molecules and also of a heat distribution (Fig. 10.1.1).

![Fig. 10.1.1. Scalar spot strain in the laminar flow (Ottino, 1989)](image)

When the rate of mixing due to heat and mass diffusion is determined with the use of the Fourier and Fick laws (Лапин, 1989), which establish a direct proportionality between
diffusive fluxes and temperature/concentration gradients, such molecular processes can be stated to be important in those flow regions where a large scalar field gradient exists.

Mixing due to molecular diffusion and heat conduction is known to proceed too slowly (Лапин, 1989; Libby, 1994). Turbulence accelerates much this process under the joint action of the mechanisms similar to those in the laminar flow and an additional diffusive transfer (turbulent diffusion) (Fig. 10.1.2) because of random turbulent fluctuations. Mixing shows up at all turbulence length and time scales – from integral to Kolmogorov scales up to Bachelor ones. The role of vortex structures with a size larger than a characteristic one of the scalar field inhomogeneity implies that unmixed volumes of a medium are convected when they do not much contribute to the mixing process. Usually, information about such a macromotion is gathered in the averaged scalar field of concentration and temperature.

![Fig. 10.1.2. Scalar spot strain due to turbulent diffusion (Fox, 2003)](image)

At turbulent motion scales small as compared with scale changes in the averaged scalar field, the irregular motion of a medium much strains the region occupied by the scalar field, whereas the heat and concentration of mixture species are propagated in the form of extremely wrinkled layers and spots. The scalar field changes in a stepwise fashion from layer to layer, as a result of which sharp temperature and concentration gradients are formed in it. So, a scalar field distribution in the flow field is generated by small-scale turbulent transfer, at which molecular diffusion and heat conduction start playing a decisive role, and in the course of mixing the interaction makes itself evident between small-scale turbulent motion, on the one hand, and molecular diffusion and heat conduction, on the other hand. The mechanism analyzed here makes a molecular contact with mixture species, and this is a necessary condition for chemical reactions to proceed. Here the interlayers of a reacted medium appear and scalar fields become more ‘wrinkled’. Moreover, the chemical reaction is also peculiar of its interaction scales: chemical reaction time, flame thickness at combustion, etc.

Thereby, mixing of scalar fields in the turbulent flow can be conventionally divided into three processes (Fox, 2003; Ionash, 1984):

- large-scale transfer by large turbulent vortices (macromixing);
• mixing up to Bachelor scales due to the small-scale fluctuation motion of a medium (Bachelor length scale $l_b = l_k/\overline{Sc}$ where $l_k$ is the Kolmogorov length scale and $Sc$ is the Schmidt number);
• mixing up to the molecular level due to molecular diffusion and heat transfer via molecular conduction.

The two last processes are defined by the properties of small-scale motions (micromixing), very often the small-scale structure is therefore considered in the local homogeneity and isotropy approximation despite the fact that the main flow and large-scale fluctuations in any real turbulent motion are inhomogeneous and anisotropic. In this case, the change in the mixture species to the homogeneous concentration state due to small-scale turbulence can be described using theory of local-homogeneous turbulence (Monin, 1967; Hinze, 1963). Its important features are the notion of cascade energy transfer through a turbulence scale spectrum and the assumption on the instability of large flow vortices and their multiple splitting into smaller ones up to the moment when vortices of such a small size are formed and a turbulent Reynolds number, $Re_t$, for this size becomes of the order of unity. As such a process is random in character and the number of splittings is great, the motion of large vortices should not influence essentially that of small ones. This conclusion lies at the basis of the Kolmogorov theory of universal equilibrium regime for local-homogeneous turbulence (Колмогоров, 1941).

10.1.3. Mixing-controlled reactive flows and flames

The generally accepted view of turbulent non-premixed reactive jets and combustion means that such phenomena are controlled basically by mixing. In the classical work (Hawthorne, 1948), the flame lengths and the mean structure of turbulent jet diffusion flames are studied and are found to correlate well with the mixing laws of turbulent jets. This view is closely related to that of laminar diffusion flames as studied by Burke and Schumann (Burke, 1928) where mixing occurs by molecular diffusion in the laminar flow. The flame is at the surface where fuel and oxidant are in the stoichiometric ratio (reaction zone). This zone is very thin. This case suggests that rates of chemical reaction are much faster than those of mixing due to molecular diffusion. This has been extended to turbulent diffusion flames. If fuel is in excess, mixture consists of products and excess fuel; if oxygen is in excess, mixture consists of products and excess oxidant. A variable known today as mixture fraction (Turns, 1996) is responsible for the mixture stoichiometry, and instantaneous temperature and species concentrations can also be related to the mixture
fraction. At some point of the turbulent reactive flow, the mixture fraction fluctuates randomly and its statistics are similar to those in the non-reactive flow. Averaged species concentration and temperature values can be obtained from their instantaneous coupling with the mixture fraction through a mixture fraction PDF. Such averaged values are indicative of the fact that fuel and oxidant co-exist. This is not attributed to the behavior of slow chemical reactions but to the fact that significant concentrations of fuel and oxidant may exist at the same place but not at the same time. Such a mixing-controlled concept arises the question of its range of validity. If the mixing process could be made fast enough, then reaction rates would be controlled by chemical properties and not by mixing.

In the early 1970s, interest was again focused on chemical kinetics because it was needed to reduce air pollution emissions (e.g., nitric oxide) due to combustion processes. Nitric oxide is formed in the course of relatively slow reactions. So, the mixing-controlled concept discussed above cannot directly apply to predict nitric oxide, and many efforts have been made to reveal a simple relationship between its production and convective/mixing time scales in the flow. Hydrogen jet diffusion flames results (Bilger, 1974) have suggested that the reactive time scale is minus the half power dependence on the Reynolds number and is proportional to the Kolmogorov time scale. In (Driscoll, 1992), it is shown that radiation and differential diffusion are also important, particularly in hydrocarbon flames, and it is difficult to find such a simple dependence.

In the 1960s and 1970s, primary emphasis was concentrated on how the mixing rate was affected not only by changes in density but by its fluctuations, and by flame-generated turbulence (Gunther, 1983). For this purpose, J.H. Kent and R.W. Bilger (Kent, 1972) studied a horizontal round-jet diffusion flame of hydrogen in an air co-flowing stream with new non-dimensional parameters such as the jet-to-co-flow velocity ratio and some boundary layer characteristics at the jet nozzle exit. This approach applies to study non-reactive jet flows. Classical work (Launder, 1972) on self-preserving and equilibrium flows was promoted by attempts to describe the turbulence by partial differential equations with regard to turbulent production, advection, diffusion, and dissipation with the use of the RANS approach. K.N.C. Bray emphasized (Bray, 1980) that many more terms are needed to describe flows with strong density fluctuations. Density-weighted or Favre averaging of the Navier–Stokes equations and transport equations of species seems to be one of the ways how to tackle the problem stated (Лапин, 1989; Libby, 1994).

The chemistry modeling technique was extended to include the concept of chemical equilibrium being fast chemistry so that temperature effects would assess intermediate species concentrations. This was particularly helpful for modeling nitric oxide. In (Bilger,
1980) dealing with hydrogen jet diffusion flames, the kinetics was assumed to be similar to that of Zeldovich’s reduced mechanism and temperature and oxygen concentrations to be given by the fast chemistry concept. Consequently, averaged chemical reaction rates, needed in the Reynolds/Favre-averaged equation for the species, would be obtained by weighting instantaneous reaction rates, derived from fast-chemistry theory, in terms of the mixture fraction PDF (Libby, 1994).

The analysis as made in (Bilger, 1980) discovered that reaction rates in fast-chemistry flames are proportional to scalar dissipation ones, or to molecular mixing ones. Scalar dissipation rates in turbulent jets discharging into quiescent air are defined by minus the fourth power of distance from the nozzle exit. Finite-rate chemistry would be most profound near the nozzle exit. The bulk of carbon monoxide burns out much further downstream where scalar dissipation rates are very low. So, reactive species concentrations and temperatures depend not just on mixture fraction, as under the fast-chemistry concept, but also on local scalar dissipation values. In (Tsuji, 1968), for laminar counter-flow diffusion flames, the composition and temperature were shown to be dependent on flow strain rate – quantity linearly related to the scalar dissipation of the mixture fraction. At high strain rates (scalar dissipation rates) temperatures decrease, and the profiles of fuel and oxidant concentrations are much overlapped. At very high strain rates, the chemical reaction is quenched.

The fast chemistry implied that chemical equilibrium does not apply at low scalar dissipation rates. Experiments showed that CO concentration levels are far below those predicted by full chemical equilibrium (Mitchel, 1980). The experimental analysis of laminar diffusion flames (Libby, 1994; Bilger, 1980) indicated that chemical equilibrium is not reached at low strain rates. Such experimental data can be adopted for turbulent flame calculations (Bilger, 1980). In (Lieuw, 1981), it was revealed that CO levels in turbulent diffusion flames can be well predicted, assuming that moderately low strain rates found from laminar flame calculations can be used. These concepts (Bilger, 1980; Lieuw, 1981) have become known as the modeling of laminar flamelets.

The concept of fast chemistry dependent on scalar dissipation rate in regions, where the mixture fraction is close to the stoichiometric one, is supported by the Sandia experiments on piloted-jet diffusion flames (TNF Workshop web site). Work (Masri, 1986) involving probe measurements included later Sandia Rayleigh/Raman scattering measurements and eventually published in (Dibble, 1987; Masri, 1988). These laser-diagnostic measurements (TNF Workshop web site; Masri, 1988; Mansour, 1988; Dally, 1998) are the experimental databases that are the current benchmark for turbulent non-premixed jet flow and
combustion modelling (TNF Workshop web site). The database now comprises measurements of all bulk concentrations and temperature for a wide range of jet and bluff-body flames (TNF Workshop web site). The high quality of the data allows modelers to refine models and theoreticians to develop new concepts.

10.1.4. State parameters of heat and mass transfer in the flow field

A mixture normalized temperature $\theta$; a species concentration $Y_\alpha$, different-kind inert admixtures and additions to the flow; specially constructed Shvab–Zeldovich variables $\beta_\alpha$ or a mixture fraction $f$ for turbulent flows with non-premixed species (Bilger, 1980); a progress variable $Z$ for mixing with premixed species (Bray, 1980) are presumed as scalar state parameters of heat and mass transfer in the flow field.

When turbulent mixing is under consideration, scalar fields are classified according to several main factors. The first factor is connected with the concepts of conserved and dissipative scalars (Libby, 1994; Fox, 2003). A criterion for a conserved scalar (denoted by $f$) is the evidence that mixing of a scalar field occurs via convective and diffusive transfer when the scalar neither disappears nor appears due to different-nature sources and sinks. Usually, the simplification of the statement of the problem on the statistics of several scalar quantities reduces to finding variables related to reactant concentration and temperature so that these new variables would become conserved (Libby, 1994; Veynante, 2002; Brodkey, 1975 (a)). In turbulent mixing studies, the selection of such variables and the formulation of transport equations coincident with them is the major way to simplify the intricate behavior of mathematical models.

The second factor characterizes scalars through their influence on the dynamic characteristics of the flow. Scalars are divided into active and passive (Brodkey, 1975 (a); Warhaft, 2000). Active scalars can essentially influence the flow velocity, which is realized at large temperature, density, concentration drops in the flow, in the course of chemical reactions with intense heat release, etc.

When passive scalars are mixed their influence is seen only in one direction – from velocity field to scalar field, i.e., an inverse relationship is assumed to be absent. This means that the transfer of the scalar field and chemical reactions occurring in it do not exert any influence on the fluid dynamics. In the case of such-flow idealization, the following mixture properties are chosen: reactant concentration is small, reaction heat is small, mixture density and kinematic viscosity values are almost constant, molecular diffusion coefficients of each of the species and the mixture are taken as equal and
invariable (Brodkey, 1975 (a); Warhaft, 2000). Such simplification is not always admissible. However, the assumptions for modeling of passive scalar are made with a sufficient accuracy, as applied to such admixtures as smoke, dust, bacteria, moisture, gases, and solutions with small admixture concentration, and call for engineering approaches adopted in studies (Brodkey, 1975 (a); Warhaft, 2000).

10.1.5. Mixture fraction concept for reactive flows

A conserved scalar is exemplified by the quantity \( f \) that depends linearly on the reactant concentration via the combination

\[
\begin{align*}
  f &= a \sum_{j=1}^{n} p_j Y_j + b
\end{align*}
\]

(Libby, 1994) where \( a, b, p_j \) are constant coefficients. Mathematically, the transport equation for mixture fraction with no regard to the source terms is

\[
\frac{\partial \rho f}{\partial t} + \frac{\partial \rho U_j f}{\partial x_j} = \frac{\partial}{\partial x_k} \left( D \frac{\partial f}{\partial x_k} \right).
\]

(10.1.1)

Such a condition is implemented, for example, during mixing of flows with different temperatures or with several chemically inert relative to each other species. As another example, let us mention here the so-called case of mixing at chemical equilibrium (Libby, 1994) when chemical reactions in the flow field proceed almost instantaneously with respect to a total mixing time, as soon as the reactant concentration in the flow field has attained a required proportion. The thermochemical state of the flow in this case is determined by mixing time and is described by one conserved scalar variable. The use of such an approximation is most prominent in the study of diffusive combustion of different media (Libby, 1994; Veynante, 2002). According to theory of conserved scalars, the mixture fraction \( f \), the total enthalpy or one of the Shvab–Zeldovich variables \( \beta_a \) can be chosen as a scalar, and the remaining thermodynamic parameters are analyzed as a function of one variable alone (Libby, 1994).

For illustration of the effectiveness of introducing conserved variables, let us consider two non-premixed volumes of medium – \( V_A \) and \( V_B \) that contain inert admixtures with different initial concentrations \( Y_{A0} \) and \( Y_{B0} \) and fill some volume \( V \) (Fig. 3).

If a relative concentration \( Y_A/Y_{A0}, \) of component \( A \) is designated through \( f \) with an entire volume-averaged quantity

\[
\bar{f} = \frac{V_A}{V_A + V_B} = \frac{V_A}{V},
\]

then a relative concentration \( Y_B/Y_{B0}, \) of component \( B \) is equal to \( 1 - \bar{f} \) with an averaged value.
1 - \( f = V_B / V_A + V_B = V_B / V \). Henceforth, mixing of inert flows with different initial species concentrations is described by one conserved quantity \( f \).

![Diagram](image)

Fig. 10.1.3. Definition of the ‘mixture fraction’ concept

The next example of the model is the mixing of two flows in the presence of chemical reactions \( \omega_\alpha t, x, \rho, Y, T \). For illustration, in the turbulent flow the reactants \( A \) and \( B \) are assumed to mix and react, followed by a subsequent formation of a product \( P \) according to the second-order irreversible reaction

\[
A + nB \xrightarrow{\omega_\alpha} sP. \tag{10.1.2}
\]

The influence of such a reaction between the reactant \( A \) and the reactant \( B \), which proceeds following the Arrhenius law and depends both on reactant density and concentration and on temperature (Libby, 1994; Fox, 2003; Brodkey, 1975 (a)), can then be represented as

\[
\omega_\alpha t, x, \rho, Y, T = -\rho K \frac{Y_A Y_B}{T} \tag{10.1.3}
\]

where \( Y_A \) and \( Y_B \) – concentrations of the reactants \( A \) and \( B \) at the point \( x \) at the time moment \( t \); \( K(T) \) – Arrhenius factor representing the reaction rate as a function of the temperature \( T \), \( K(T) = k_1 T^{k_2} \exp(T_a / T) \) \( (k_1, k_2, T_a \) are the reaction constants). The factor \( K(T) \) is found in terms of a \( T \) value at the same space-time point just as the concentration of reactants and is constant in the isothermal flow. The rate \( \omega_\alpha \) is determined by (10.1.3) where by convention, one mole of the reactant \( A \) enters into the reaction with \( n \) moles of the reactant \( B \). According to (10.1.3), for the chemical reaction rate to be increased essentially, at a definite time moment in the considerable part of the flow region in question the concentrations \( Y_A \) and \( Y_B \) must not be equal to zero at the same space point, i.e., the reactants \( Y_A \) and \( Y_B \) must be blended into a sufficiently homogeneous mixture. In this case, the mixture fraction can be assigned as
Such a scalar is conserved and obeys (10.1.1) with a definition domain \( f \in [0,1] \) where \( f = 0 \) in the flow of the pure reactant \( B \) and \( f = 1 \) in the flow of the pure reactant \( A \) (Fig. 10.1.3).

### 10.1.6. Progress variable concept for reactive flows

Theory of conserved scalars (Libby, 1994) is very informative in studying most of the problems of turbulent mixing with chemical reactions and also in the presence of dissipative thermal-chemical fields. For a dissipative scalar, the dynamic system is assumed to have sources and sinks favoring an increase or a decrease in the total amount of the scalar. The chemical reaction, which causes the content of reactive species to vary in the flow field, is a good case in point. When turbulent combustion of a premixed mixture is considered, the analysis is based on a variable \( Z \) called the progress variable (Bray, 1980).

Linear relation (10.1.4) permits identifying a chemical system not in terms of three concentrations \( Y_A, Y_B, \) and \( Y_P \) but in terms of two quantities – mixture fraction \( f \) and progress variable \( Z \). The first of these quantities can be governed by the statistical approaches adopted for studying the turbulent motion of inert media. The progress variable \( Z \) characterizes the concentration of reaction products to be formed. The concentration of mixture species is determined as follows (Fox, 2003):

\[
Y_A / Y_{A0} = f - f_{st} Z \quad \text{mixing} \quad Y_B / Y_{B0} = 1 - f - (1 - f_{st}) Z \quad \text{chemical reaction} \quad Y_P / Y_{A0} = s f_{st} Z, \quad \text{mixing} \quad \text{chemical reaction}
\]

(10.1.5)

where \( f_{st} \) – stoichiometric value of the mixture fraction calculated from the condition, obeying which the chemical reaction usually occurs for the stoichiometric ratio of the reactants \( A \) and \( B \) (the equality \( n Y_A - Y_B = 0 \) is satisfied), \( f_{st} = Y_{B0} / n Y_{A0} + Y_{B0} \). Thereby, the reaction proceeds at mixture fraction values equal to the stoichiometric value of \( f_{st} \). In view of the nonlinearity of (10.1.3), the chemical reaction rate decreases essentially as deviated from the stoichiometry conditions. In addition, the definitions given in (10.1.5) illustrate how each of the variables is responsible for individual processes proceeding in the flow field. The transport equation for \( Z \) takes into account the terms for chemical sources \( \omega_Z \):
Since the model equations for dissipative thermal-chemical fields because of the presence of chemical source terms usually appear to be drastic and give rise to many difficulties for numerical realization, researchers are trying to select conserved variables of the problem that depend linearly on dissipative variables. In particular, turbulent micromixing is being investigated invoking the characteristics of the scalar field of a conserved variable, for example, the mixture fraction \( f \). For the behavior of dissipative scalars to be analyzed, conserved variable-dependent characteristics must be adopted: conditional concentration of reactants, conditional scalar dissipation rate, surface density with equal concentration (in combustion theory – flame surface density function), etc. (Veynante, 2002).

10.1.7. **Problem in calculating averaged chemical reaction rate**

The non-averaged expression \( \bar{\omega}_\alpha \ t, x_i, \rho, Y_k, T \) is determined through values of all possible thermodynamic quantities of mixture at some spatial point in the flow and at a definite time moment. Therefore, it is usually assumed that the form of the averaged (by Reynolds/Favre or by the LES) expression \( \bar{\omega}_\alpha \ t, x_i, \rho, Y_k, T \) must contain some one-point characteristics of averaged and fluctuation fields \( \rho, Y_k, T \), etc.:

\[
\bar{\omega}_\alpha (t, x_i, \rho, T, Y_{\beta}) = W_\alpha (t, x_i, \bar{\rho}, \bar{T}, \bar{\rho}^2 Y_{\beta}^2, \bar{\rho}^2 \bar{\theta}^2, \bar{\rho}^2 \bar{\theta}^n, \bar{\rho}^n \bar{\theta}^n, \bar{\rho}^n \bar{\theta}^n, ...,)
\]

(10.1.7)

where \( \rho' \) – fluctuation density; and \( \theta'' \) – temperature fluctuation in Farve averaging. Sometimes (10.1.7) is assumed to depend only on second-order correlations of turbulence characteristics. But even in this case, a number of capabilities arise how to model the chemical reaction influence on scalar field changes. Consider the simplest case of the isothermal turbulent flow of an incompressible medium with the density \( \rho = \bar{\rho} = \text{const} \) and the temperature \( T = \bar{T} = \text{const} \). Heat capacities and molecular diffusivities of mixture species are constant, species to be mixed are passive in their influence on velocity variations. In such an approximation it is sufficient to consider the Reynolds-averaged quantities. For (10.1.7) with regard to definition (10.1.3) one then arrives at

\[
\bar{\omega}_\alpha \ \bar{\rho}, \bar{T}, Y_A, Y_B = -\bar{\rho} K \ \bar{T} \ \bar{Y}_A \bar{Y}_B + \bar{y}_A \bar{y}_B .
\]

(10.1.8)
Here $y'_A$ and $y'_B$ are the reactant concentration fluctuations, $y'_A = Y_A - \overline{Y_A}$ and $y'_B = Y_B - \overline{Y_B}$. The Arrhenius factor $K \bar{T}$ is a constant quantity.

Variants of approximate expression (10.1.8) for $\bar{\omega}_a \bar{\rho}, \bar{T}, Y_A, Y_B$ are possible:

1) $\bar{\omega}_a \bar{\rho}, \bar{T}, Y_A, Y_B = -\bar{\rho}K \bar{T} \bar{Y}_A \bar{Y}_B$ is the approximation which is called the 'laminar chemistry' approximation and is valid for laminar or turbulent flows where the rate of the chemical reaction is slow in comparison with that of decay of reactant concentration fluctuations. For turbulent flows, this fact signifies that the complete mixing state (fluctuations become negligibly small) is attained well before the chemical reaction has ceased. The quantities $\overline{Y_A}$ and $\overline{Y_B}$ are calculated from the ordinary equations for averaged quantities. When the time of the chemical reaction is comparable with that of turbulent mixing such an approximation almost fully ignores the influence of turbulent fluctuations on the chemical reaction rate but these fluctuations are still taken into account in some fashion in calculating $\overline{Y_A}$ and $\overline{Y_B}$. The correlation $\overline{y'_A y'_B}$ in (10.1.8) is responsible for such an influence and should be determined somehow.

2) $\bar{\omega}_a \bar{\rho}, \bar{T}, Y_A, Y_B = -\bar{\rho}K \bar{T} \bar{Y}_A \bar{Y}_B + \overline{y'_A y'_B}$ is an exact relation. The problem consists in obtaining $\overline{y'_A y'_B}$ and is being solved by deriving and closing the equation coincident with this correlation for the reactant concentration.

3) $\bar{\omega}_a (\bar{\rho}, \bar{T}, Y_A, Y_B) = \iint \omega_a (\bar{\rho}, \bar{T}, \Gamma_A, \Gamma_B) P(x_i, t, \Gamma_A, \Gamma_B) d\Gamma_A d\Gamma_B$ (10.1.9)

where $\Gamma_A$ and $\Gamma_B$—probability variables of the reactant concentrations $Y_A$ and $Y_B$, respectively. According to (10.1.9), there is no need to build up approximate expressions for $\bar{\rho}$, if one adopts the notion of the joint PDF $P(x_i, t, \Gamma_A, \Gamma_B)$ of random quantities. Such an approach permits obtaining an expression for $\bar{\omega}_a$ with allowance for all concentration correlations. Nevertheless, one has to ‘invent’ a form of multi-dimensional PDFs for real turbulent flows invoking more or less realistic physical concepts or to formulate and solve the corresponding differential equations. In the case when one restricts himself to the ‘laminar chemistry’ approximation, i.e., the equality $\bar{\omega}_a \bar{\rho}, \bar{T}, Y_A, Y_B = -\bar{\rho}K \bar{T} \bar{Y}_A \bar{Y}_B$ is written, the simplest form of the PDF
\[ P(x_i, t, Y_k) = \prod_{k=1}^{n} \delta(Y_k - \bar{Y}_k) \] is mainly used. Of late, the mathematical models for different turbulent reactive flows, which contain systems of equations for statistical moments, have then found use along with the probability methods (Fig. 10.1.4) (Libby, 1994; Fox, 2003; Veynante, 2002).

From the above-said, it follows that the concept of the mixture fraction \( f \) can be applicable for description of inert species mixing at \( Z = 0 \), i.e., for one of the limiting cases when the Damköhler number (Da), establishing the interaction between turbulent mixing characteristic time scales \( \tau_t \) and the chemical reaction \( \tau_{ch} \), is equal to zero.

Theory of conserved scalars is demanded for another limiting case of an infinite fast reaction (Da → +∞), assuming that the chemical reaction rate is limited by the mixing rate of reactants. Such an assumption is attractive for practically important cases as many reactions have fast rates and are considered to proceed instantaneously when reactants appear in the stoichiometric ratio. Quantitatively, the fast chemical reaction can be expressed by the approximation \( Y_A x_i, t Y_B x_i, t = 0 \), i.e., two reactants do not exist simultaneously at one point and at the same time moment since such a reaction cancels them. In this case, the averaged rate \( \bar{\omega}_A \) cannot be calculated by (10.1.8) and for this purpose, different approaches should be adopted. One of them is the PDF method, where the problem reduces to calculating integral (10.1.9), if the joint PDF \( P x_i, t, \Gamma_A, \Gamma_B \) of reactant concentration is assigned somehow. An important consequence of the assumption on an infinite fast chemical reaction is that an instantaneous reactant concentration \( Y_a = Y_a f \) is a function of only one conserved variable – mixture fraction \( f \). In this case, the task of defining the interaction between turbulence and chemical processes is much simplified since the statistics of all chemical variables can be derived from the knowledge of mixture fraction statistics.

For the chemical reaction, at Da → +∞ the reactants A and B are completely separated in the flow. The reactant B (\( Y_{B0} = 0 \)) is absent in the regions where \( f > f_{st} \), and, vice versa, the reactant A (\( Y_{A0} = 0 \)) is absent in the regions where \( f < f_{st} \), because as soon as reactants have been mixed in a proportion required by the reaction, they react instantaneously and completely. The instantaneous values of reactant concentration (10.1.5) and progress variable \( Z \) then build up linear relationships (Fig. 10.1.4):

\[ a) \text{ at } f < f_{st} \quad Z_\infty = \frac{f}{f_{st}}, \quad \frac{Y_A}{Y_{A0}} = 0, \quad \frac{Y_B}{Y_{B0}} = \frac{1-f}{f_{st}}, \quad \frac{Y_p}{Y_{A0}} = sf^*, \]
b) at $f_{st} Z_{\infty} = \frac{1-f}{1-f_{st}}, \frac{Y_A}{Y_{A0}} = \frac{f-f_{st}}{1-f_{st}}, \frac{Y_B}{Y_{B0}} = 0, \frac{Y_P}{Y_{A0}} = \frac{s f}{1-f_{st}}$.

Henceforth, in the two limiting cases ($Da = 0$ and $Da \to +\infty$), the mixture state can be characterized by one scalar variable $f$ since for mixing of inert species $Z = 0$ and for mixing with an infinite fast chemical reaction $Z = \min\left(\frac{f}{f_{st}}, \frac{1-f}{1-f_{st}}\right)$. Within the framework of such approaches, the influence of turbulent fluctuations on statistical parameters is allowed for by the PDF $P(x_i, t, \Gamma)$ of the mixture fraction $f$, whereas averaged concentrations are $\bar{Y}_\alpha = \int_0^1 Y_\alpha(\Gamma) P(x_i, t, \Gamma) d\Gamma$ where $\Gamma$ is the probability variable of $f$.

![Diagram](image)

Fig. 10.1.4. Instantaneous values of reactant concentration and progress variable vs. mixture fraction at $Da \to +\infty (n = 1, s = 2, f_{st} = 0.4)$

The model for turbulent mixing with the chemical reaction can be simplified using the concept of the mixture fraction $f$ from (10.1.4), for which transport equation (10.1.1) has no chemical source terms. In this case, the description of the mixing process reduces to the analysis of changes in the mixture fraction and its statistical moments independent of $Da$.

### 10.2. Modeling approaches of fluid dynamics

To describe the motion of a continuous medium, one uses the Euler (random fields are related to a fixed coordinate system) or Lagrange (transfer and thermal-chemical behavior
of mixture are assigned by the properties of a set of moving medium particles) approach (Лапин, 1989). No matter which of these approaches is taken for building up mathematical models and with which of the above-mentioned statistical characteristics one has to deal, the classical conservation equations (Navier – Stokes equations, continuity equation, corresponding transport equations for mixture species temperature and concentration, thermodynamic state equations) are usually assumed to serve as the starting point for an exact mathematical representation of convective and diffusive transport processes, chemical reactions, and the action of body forces. Under the appropriate initial and boundary conditions, these equations govern the dynamics of flow and from their solution subject to the initial and boundary conditions one can determine individual realizations of random fluid-and thermodynamic fields in the flow.

For the multi-component single-phase mixture flow with chemical reacting, at which radiation is considered to be negligibly small, the variation in the flow velocity $U_i$ is determined by:

continuity equation:

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho U_i}{\partial x_i} = 0,$$

(10.2.1)

the Navier – Stokes equations for velocity components $U_i(x_j, t)$

$$\frac{\partial \rho U_i}{\partial t} + \frac{\partial \rho U_j U_j}{\partial x_j} = -\frac{\partial P}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} + F_i,$$

(10.2.2)

where $\tau_{ij}$ – viscous tensor, $\tau_{ij} = \rho \nu \left[ \left( \frac{\partial U_j}{\partial x_j} + \frac{\partial U_i}{\partial x_i} \right) - \frac{2}{3} \frac{\partial U_k}{\partial x_k} \delta_{ij} \right]$; $\nu$ – kinematic viscosity; $F_i$ – source of body forces. Hereinafter, the summation is performed over repeating indices.

Transfer equations for concentrations $Y_{\alpha}(x_j, t)$ (concentration of reactants, reaction products, inert admixtures, etc.) are of the form

$$\frac{\partial \rho Y_{\alpha}}{\partial t} + \frac{\partial \rho U_j Y_{\alpha}}{\partial x_j} = -\frac{\partial J_{\alpha}^j}{\partial x_j} + \omega_{\alpha}, \quad \alpha = 1, \ldots, N,$$

(10.2.3)

$$\sum_{\alpha=1}^{N} Y_{\alpha}(x_j, t) = 1.$$

(10.2.4)
Since the diffusive effects are not so significant in the problems on turbulent mixing with chemical reacting (Лапин, 1989; Libby, 1994; Fox, 2003; Veynante, 2002), the processes of heat and mass transfer as a rule are regarded under the assumptions essentially simplifying the mathematical problem statement. Usually the molecular flow $J^{\text{m}}_j$ is approximated with a sufficient accuracy by Fick’s law. Diffusivity coefficients of mixture species are assumed to be equal (Libby, 1994) owing to the fact that in turbulent flows with large Re numbers the effects dependent on different values of diffusivity coefficient are ignored when using the hypotheses similar to Re number similarity ones. A consequence of this is the elimination of cross diffusive effects from the consideration.

Write an equation for the enthalpy, $H(x_i, t)$, of a mixture:

$$\frac{\partial \rho H}{\partial t} + \frac{\partial \rho U_j H}{\partial x_j} = \frac{\partial p}{\partial t} - \frac{\partial J^H_j}{\partial x_j}$$

(10.2.5)

where $J^H_j$ – heat flux with regard to the assumption on the validity of Fourier’s law

$$J^H_j = -\frac{\rho v}{Sc} \left[ \text{Le} \frac{\partial H}{\partial x_j} + (1 - \text{Le}) \sum_{\alpha=1}^{N} h_{\alpha} \frac{\partial Y_\alpha}{\partial x_j} \right].$$

In the general case, the coefficients reflecting the thermodynamic properties of a medium, i.e., $D$, $\nu$, $\lambda$, etc. are the functions of mixture temperature and concentration composition. The Schmidt number (Sc) is a measure of the relative importance of the viscous and diffusive properties of a medium. For gases and gas mixtures this parameter as well as the Prandtl number (Pr) is often taken equal to unity although its value can essentially differ from unity, for example for binary mixtures of gases with large and small molecular weights. More often the ratio of these two parameters – the Lewis number (Le) – is assumed to be equal to unity. Then equation (10.2.5) for the enthalpy $H(x_i, t)$ is still more simplified because the second summand disappears from the writing of the heat flux: $J^H_j = -\frac{\rho v}{Sc} \frac{\partial H}{\partial x_j}$. Further, let us restrict our consideration to small Mach numbers (M), which allows equation (10.2.5) to be simplified since the term $\frac{\partial p}{\partial t}$ responsible for the propagation of acoustic waves disappears. In equation (10.2.5) the kinetic energy and viscous dissipation as well as the work due to the action of body forces are not allowed for:
\[
\frac{\partial \rho H}{\partial t} + \frac{\partial \rho U_j H}{\partial x_j} = \frac{\partial}{\partial x_j} \left( \rho v \frac{\partial H}{\partial x_j} \right). \tag{10.2.6}
\]

System (10.2.1)–(10.2.6) consists of \( N + 5 \) equations that contain \( N + 6 \) unknown quantities. In the case of gas flows, for the system to be closed, the state equation for a perfect gas \( p = \rho RT \sum_{\alpha=1}^{N} \left( Y_{\alpha} / M_{\alpha} \right) = \rho RT / M \) is usually adopted. Here \( M_{\alpha} \) is the molar mass of the \( \alpha \)-species, \( M \) is the molar mass of a mixture, \( M = 1 / \sum_{\alpha=1}^{N} \left( Y_{\alpha} / M_{\alpha} \right) \). For liquid, it is possible to consider the polynomial density dependence on temperature \( \rho = \rho_0 + AT + BT^2 \). Here \( A \) and \( B \) are constant coefficients. Later expressions are additional thermodynamic relations in order to build up a relationship between the enthalpy of each of the species and the medium temperature: \( H = \sum_{\alpha=1}^{N} h_{\alpha} Y_{\alpha} \). The enthalpy of the \( \alpha \)-species is approximated as \( h_{\alpha} = h_{\alpha}(T) = \int_0^T c_{p,\alpha}(\phi) d\phi = c_{p,\alpha} T + b_{\alpha} \) with the use of two constants \( c_{p,\alpha}, b_{\alpha} \), which permits one to write a definition for enthalpy as \( H = C_p T + \sum_{\alpha=1}^{N} b_{\alpha} Y_{\alpha} \) where \( C_p \) is the specific heat capacity of a mixture, \( C_p = \sum_{\alpha=1}^{N} Y_{\alpha} c_{p,\alpha} \).

An alternative approach to equation (10.2.6) for enthalpy may be an equation for temperature. At small values of the Mach number it looks like:

\[
\frac{\partial \rho C_p T}{\partial t} + \frac{\partial \rho C_p U_j T}{\partial x_j} = \frac{\partial}{\partial x_j} \left( \lambda_{j} \frac{\partial T}{\partial x_j} \right) + \omega_T + \rho D \frac{\partial T}{\partial x_k} \sum_{\alpha=1}^{N} c_{p,\alpha} \frac{\partial Y_{\alpha}}{\partial x_k}. \tag{10.2.7}
\]

where \( \omega_T = -\sum_{\alpha=1}^{N} h_{\alpha} \omega_{\alpha} \). Parameters requiring their definition in the equations for species concentration (10.2.3) and temperature (10.2.7) are represented as the quantities \( \omega_{\alpha} \) being functions of different-nature sources or sinks. Usually they depend intricately on the thermophysical parameters of the flow and in the general case are strongly non-linear with respect to these quantities. In the case of chemical reaction, \( \omega_{\alpha}(\rho, Y_j, T) \) is its rate per unit volume of the mixture \( \alpha \)-species with respect to other species that depends on the kinetics of flow chemical reactions of different nature and complexity.
Thus, by solving the system of equations (10.2.1)–(10.2.7) under the initial and boundary conditions one can determine individual realizations of random fluid and thermodynamic fields in the flow.

10.2.1. Direct Numerical Simulation

Advances in computational technologies have permitted one to directly integrate transport equations for random quantities and have resulted in developing the method of Direct Numerical Simulation (DNS) (Moin, 1998) (Fig. 10.2.1). Further information is being processed by the methods of statistical analysis.

In the DNS method, the Navier–Stokes equations and the transport equations for scalars (10.2.1)–(10.2.7) are being solved numerically using the pseudo-spectral approach (Eswaran, 1988), finite-difference methods (Shumann, 1984), and the finite-element method (Brethouwer, 2000) on a computational grid with cells, whose size is comparable with the smallest scale of possible fluid-thermodynamic length scales (Kolmogorov scale). Although the DNS requires considerable CPU time and applies only to simplest-geometry systems, nothing other than it yields detailed information about the structure of fluid and thermodynamic flow fields. Advances of such an approach are outlined elsewhere in (Moin, 1998; Shumann, 1984; Brethouwer, 2000; Sripakagorn, 2000). At the same time, on the way of its wide application an insuperable barrier is created by the insufficient power of modern computers. DNS limitations are associated with a great many nodal
points of a 3D computational grid even at moderate Reynolds, Pecklet, Schmidt, and Damköhler numbers.

As an alternative approach to the DNS are the methods that are based on averaging the realization equations such as the LES method (Ferziger, 1987) and statistical models for fluid-and thermodynamic parameters averaged by Reynolds/Favre (Лапин, 1989; Libby, 1994; Fox, 2003; Коловандин, 1980) (Fig. 10.2.1).

10.2.2. Large Eddy Simulation

In the LES-based approach, anisotropic large-scale vortices with a scale exceeding a size of a finite-difference cell are exactly reproduced by numerical integration of realization equations for flow characteristics, whereas small-scale vortices are modeled either by simple gradient relations (Shumann, 1984) or by specially developed dynamic SGS models (Germano, 1991). Vortices are split into small and large scales by spatial filtration of realization equations where the subgrid stresses and the terms for the influence of small scales on mixture species transfer and chemical reaction in a dynamic system should be parameterized using the SGS models (Ferziger, 1987; Veynante, 2002; Germano, 1991). In this case, there appear difficulties similar to those in closing the statistical models for the quantities averaged by Reynolds/Favre. Unlike the DNS approach, the LES method requires that unknown quantities related to chemical reactions be calculated. In this case, the accuracy of proposed results depends on the quality of such calculation. In problems of turbulent transfer with chemical reaction at large Da numbers, the thickness of the reaction front is usually small, the averaging over small length scales is therefore very difficult to perform coincident with the physics of the process considered.

10.2.3. Reynolds-averaged Navier – Stokes approach

The models for statistical moments of fluid-thermodynamic quantities averaged by Reynolds (Лапин, 1989; Libby, 1994; Fox, 2003; Коловандин, 1980) are most open to numerical realization. For a variable density, usually one uses Favre averaging (Libby, 1994) related to Reynolds averaging: $\bar{q} = \bar{\rho} + \bar{\rho'} q' / \bar{\rho}$. Here, the symbol $'$ means Reynolds averaging, $\sim$ – Favre averaging, and $\rho'$, $q'$ are the Reynolds fluctuations of the density and the quantity to be calculated, $\rho' = \rho - \bar{\rho}$, $q' = q - \bar{q}$. In this case, the structure of the equations for averaged quantities remains almost such as for a constant density.
Not being restricted to CPU time, the statistical models have essentially other severities. Since the equations for turbulence statistical characteristics are derived from nonlinear differential equations for velocity, temperature, and concentration, an unclosed set of equations with their boundless great number is formed. Limiting the sequences of such equations requires that the phenomenological and statistical hypotheses be applied, and this is the essence of the turbulence closure problem (Лапин, 1989; Коловандин, 1980; Монин, 1967).

In an attempt to close equations for averaged turbulence characteristics of concentration and temperature from the corresponding transport equations and to solve the general problem of closing the equations with regard to the chemical reaction terms, one meets the necessity to average the terms $\omega_\alpha \ t, x_i, \rho, Y_k, T$ responsible for the chemical reaction. For example, for a variable density, let us consider instantaneous values of the concentration $Y_\alpha$ ($\alpha = 1, \ldots, N$) (just as vector components of the flow velocity $U_i$) in the form of the sum of the Farve-averaged quantity $\bar{Y}_\alpha = \frac{1}{T} \int_0^T Y_\alpha(t) dt$ and its fluctuations $y''_\alpha = Y_\alpha - \bar{Y}_\alpha$ ($\bar{U}_i = \frac{1}{T} \int_0^T U_i(t) dt$ and $u''_i = U_i - \bar{U}_i$, respectively). Averaging of the transport equation for concentration yields the relation (Libby, 1994)

$$\frac{\partial \bar{p} \bar{Y}_\alpha}{\partial t} + \frac{\partial \bar{p} \bar{U}_i \bar{Y}_\alpha}{\partial x_j} = \frac{\partial}{\partial x_k} \left( \rho \bar{D} \frac{\partial \bar{Y}_\alpha}{\partial x_k} \right) - \frac{\partial \bar{p} u''_j y''_\alpha}{\partial x_j} + \bar{\omega}_\alpha. \tag{10.2.8}$$

The above equation contains the unclosed terms for the turbulent flow $\bar{p} u''_j y''_\alpha$ and the averaged chemical reaction rate $\bar{\omega}_\alpha \ t, x_i, \rho, Y_k, T$. The closure procedure lies in obtaining expressions for these terms through the averaged characteristics, for which the equations are built up. For velocity components, it is possible to write equations similar to the Reynolds ones. The remaining realization equations are also built up, obeying Farve averaging (Libby, 1994).

In the statistical models for inert turbulent flows, the equations are more often written for statistical moments of lower order – unknown averaged quantities, root-mean-square fluctuations, and scale-type characteristics or dissipation rate (Коловандин, 1980; Монин, 1967; Brodkey, 1975 (a)). In this case, statistical moments of definite type are expressed in terms of those of lower order or averaged flow characteristics. Among such models are those which have already become classical: the Kolmogorov – Prandtl model with the equation for turbulence kinetic energy, the two-parameter turbulence $k – \varepsilon$ model,
as well as models with the closure of the equations for Reynolds stresses and moments of higher order. Similarly, closure relations are built up within the framework of the method of statistical moments for reactive media (Libby, 1994; Fox, 2003; Veynante, 2002).
11. Modern software for fluid dynamics problems

Andrei Chorny, Yulia Zhukova
Belarusian State University, Physical Faculty
**Introduction**

Numerical simulation of gasdynamic processes is based both on the rapid progress, in particular of personal computers (PC) and on the improvement of applied software where the basic principles of computational fluid dynamics (CFD), specifically of gasdynamics are realized. Over the last several decades CFD software has gone the way from primitive scientific programs in the form of separate codes written in different programming languages (FORTRAN, ALGOL, PASCAL, C, etc.) to universal and specialized codes of applied programs. This fact is the result of a wider and successful use of CFD software in different fields:

<table>
<thead>
<tr>
<th>Aerospace:</th>
<th>Chemical Processes:</th>
</tr>
</thead>
<tbody>
<tr>
<td>• cooling of gas turbine blades;</td>
<td>• mixers;</td>
</tr>
<tr>
<td>• gas flow in turbine cavities;</td>
<td>• jet dryers and reactors;</td>
</tr>
<tr>
<td>• rocket combustion chambers;</td>
<td>• porous filler reactors;</td>
</tr>
<tr>
<td>• flame modeling</td>
<td>• furnaces and ovens</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Automobile:</th>
<th>Mechanical:</th>
</tr>
</thead>
<tbody>
<tr>
<td>• external aerodynamics;</td>
<td>• centrifugal pumps;</td>
</tr>
<tr>
<td>• air movement under the motor</td>
<td>• heat exchangers;</td>
</tr>
<tr>
<td>hood;</td>
<td>• recuperators</td>
</tr>
<tr>
<td>• cooling system in the car;</td>
<td>• flowrate meters and pipelines;</td>
</tr>
<tr>
<td>• combustion process in the</td>
<td>• vapor turbine blades</td>
</tr>
<tr>
<td>internal combustion engine</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Buildings and Environment:</th>
<th>Other Fields:</th>
</tr>
</thead>
<tbody>
<tr>
<td>• air heating and conditioning;</td>
<td>• oil-field exploration;</td>
</tr>
<tr>
<td>• wind loading;</td>
<td>• nuclear engineering;</td>
</tr>
<tr>
<td>• ventilation;</td>
<td>• energy production;</td>
</tr>
<tr>
<td>• fire safety</td>
<td>• pharmaceutics;</td>
</tr>
<tr>
<td></td>
<td>• physiological solutions;</td>
</tr>
<tr>
<td></td>
<td>• electronic installation cooling</td>
</tr>
</tbody>
</table>

To date, the feasibility of relatively inexpensive WorkStations, as well as of more widespread and available PCs enables one to model complex gasdynamic processes within a reasonable time. This makes the modeling itself attractive for solution of both fundamental and practically important problems associated with developing new technologies or forecasting phenomena in nature. The intensive development of CFD software over several last decades is attributed not only to the computing progress, but also to the elaboration of physical-mathematical models (turbulence, combustion, radiant heat...
transfer, etc.) and efficient computational methods. In solving CFD problems such properties of applied software provide physicist-researchers with:

- obtaining data that are difficult to have from experiments;
- achieving accuracy and information content of parameters of moving media and processes that occur in them;
- comparing experimental and predicted data in the course of research;
- realizing the control principles of the solution selection at each modeling stage;
- reducing the research time and cost.

Today, a universal, specialized software is represented by multifunctional systems written, as a rule, in object-oriented programming languages and having the “triad” structure: preprocessor with mesh builder - solver – postprocessor with GUI. The software structure usually comprises a number of physical-mathematical models for description of fluid dynamics. This permits one to state a particular problem to be solved (Fig. 11.1.1).

Figure 11.1.1. Generally accepted CFD software structure

However often the computation of complex problems takes too much time and this substantially restricts the CFD software use. That is why the CFD history illustrates the development of two trends in the field of developing more perfect software. On the one
side, the progress is in the direction of improving the processor performance, which is manifested itself in one-processor supercomputers based on new electronics. On the other side, the concept of computation parallelling develops and now it is used in multicore processors, as well as in multiprocessor systems (high-performance computing systems). Each of the trends has its advantages. However if earlier parallel software was not too accessible and a very narrow circle of researchers adopted it to solve problems, then at present parallel computations become widespread due to a wide application of “desk-top“ Workstations. Owing to this, parallel software becomes necessary. It should be noted that the computational efficiency in this case permits one to compute three-dimensional unsteady flows, whose modeling on one-processor computers took about several months but now it needs several days.

This chapter presents the analysis of the modeling features of hydro/gasdynamic processes, the basic elements of CFD software and the trends of its development. Several examples of the most popular applied CFD software (PHOENICS, ANSYS Fluent and STAR-CD) are considered. Rather a limited choice of examples is due to the fact that in principle, the methods for construction of such software are generally accepted at present. Usually, having understood the logic of solving CFD problems with the use of one software, a user can realize without any difficulties his solution using another. Differences in the used software lie in their internal properties that are often the result of the preferences of the authors’ software in the course of its implementation. Often they are usually of no interest for the user in comparison with the proposed result of solving the eigenvalue problem or are not available (often such software is a “black box”).

11.1. Modeling of fluid dynamics: main challenges and ways to overcome them

11.1.1. Computational domain in numerical models

The numerical solving of the problems on fluid dynamics, heat and mass transfer and other accompanying processes can be started when the laws controlling these processes are expressed mathematically, usually in the form of differential equations. Each of these equations defines a certain law of conservation of mass, momentum, and energy. In each equation some physical quantity is used as a dependent variable and the balance between different factors influencing the change in this variable is specified. Usually, in these
differential equations dependent variables are represented by the specific properties, i.e. by the properties per unit mass (density, velocity vector components, energy, species concentration, etc.).

As the differential equation is non-linear, it is impossible to obtain a solution at each point of the computational domain. So, we restrict our consideration to some set of points of this space that are called the mesh nodes or the node points. Such an approach enables one to replace the differential equation responsible for the physical process by the system of algebraic equations that is solved simply. A set of variable values at these nodes defines the numerical solution of the differential equation.

It should be noted that in fluid dynamics the model of material particle, i.e., of “infinitely small particle” (Липанов, 2001) is replaced the model of a “large particle” of finite size (volume). The space is therefore partitioned into cells or, as it is said, the medium is discretized (Roache, 1976). At this stage, there appear the problems associated with describing and presenting parameters inside a cell and those with describing cell interactions. It should be noted that when solving gasdynamic problems the inaccuracy in discretizing the differential equation (the replacement by finite-difference analogs) can change not only the quantitative measure (accuracy), but also the qualitative behavior of solutions. Minor changes, for example, of the form of finite-difference approximations, iteration schemes or the interpretation of boundary conditions can give a gain in the rate of obtaining unknown solutions or lead to catastrophic errors (Андерсон, 1990). Therefore, one should be aware of the fact that the discrete model of moving medium can substantially differ from the continuum one used for making both the differential equation and the natural phenomenon itself, whose features are of interest. Undoubtedly, the evidence of the numerical solution should be considered fair only after it has been confirmed in practice.

11.1.2. CFD ways of solving simulation problems

When discretizing the space much depends on how well a researcher “gusses” the space topology of filling the entire space with individual cells. It is obvious that the greater number of these particles, the closer the values of parameters responsible for the process under consideration can be chosen. However fluid dynamics problems have such physical phenomena that cannot be taken into account properly only by refining the space subjected to discretization because the localization scale of particles is small and their position changes in space. Among them is the turbulence phenomenon and the vortex formation in flows, the front or surface phenomenon, including the boundary layer phenomenon, etc.
These phenomena can exert a determining influence on the large-scale properties of the medium motion. At the same time, these must be resolved on an inaccessible scale level, especially at vortex formation places, and are often described only on the molecular level. Originally, these problems have manifested themselves in CFD when flow parameters are determined near surfaces (walls) and when mathematically; these have the problems of interpreting boundary conditions (Патанкар, 1984).

This problem was solved drastically with the implication of the ideas and procedures adopted in the mixed (semi-analytical or semi-discrete, differential finite-difference) methods and in the finite-element method (FEM), in particular with the use of a nonuniform distribution of parameters over a “large” particle volume and with the reduction of partial derivative equations to the solution of a system of ordinary differential equations (Седов, 1987).

As a result, only the replacement of the piecewise constant distribution of parameters in cells by the piecewise linear one enabled one to improve the approximation accuracy of solutions from the first order to the second both for space and time coordinates, \( O \Delta t^2, \Delta x^2 \). In fact, this meant the move away from the pure discrete (mesh) representation of data to local-continuity functions (splines).

Another important result is concerned with developing new (subgrid) computation algorithms of physical boundary phenomena. Their description is represented by modern know-how of software developers and computer specialists.

At present, the most promising direction in solving such problems is to apply high-performance multiprocessor computation systems based on the distributed memory architecture and to develop perfect algorithms and mathematical software that allow their efficient use. Name the basic requirements for such algorithms (Chetverushin, 2002):

- **Availability of internal parallelism.** A considered problem is partitioned into a class of sub-tasks, each of which is processed on an individual processor. Information interchange is possible during computations and at their completion in effort to obtain the final result. Such a partition can be made manually or automatically with the use of the appropriate software.
- **Providing a uniform loading of processors.** Some processors with nonuniform loading spend more time for computations as against others. Just the “slow” operation of such processors defines the final efficiency of the entire system. The uniform loading of processors is associated with the internal parallelism of algorithm. The
partition of the problem into individual sub-tasks makes sense only when the computation volume in each of them differs from each other inconsiderably.

- **Minimizing information interchanges between processors.** When the interchanged information volume is large the distributed memory system spends a considerable part of its time not for computations within the processors, but for information interchanges via relatively slow communication channels. The minimization of information interchanges further restricts the class of permissible numerical methods or it must be carried out at a time with the internal parallelism of algorithm.

- **Logic simplicity of algorithm.** This requirement is associated with the continuing progress of multiprocessor computer engineering that is accompanied by the inevitable changes in mathematical software. Simpler algorithms allow a program for a particular computation system to be written not only rapidly but also to be adapted rapidly to the next generations of multiprocessor computers.

- **Providing correct results of high-accuracy computations** (on more detailed space-time meshes).

### 11.1.3. Building the computational meshes. Mesh builders

The distribution of mesh nodes in the computation domain of fluid flow is one of the most important factors responsible for the accuracy and computational efficiency of solving the problem (Kondrashov, 2002). The concepts of building meshes are divided into one- and multiblock, component-adaptive overlapping, component-adaptive interface approaches. With the procedure of placing nodes, the meshes can be algebraic and differential (Kutler, 1985).

Among the simplest meshes are one-block meshes when the spatial computational domain is transformed to a cube. Within the framework of the one-block concept, if a cylindrical mesh is considered, only the symmetry axis is the singularity line. Many singularity lines and surfaces are inherent in the multiblock concept. So, special procedures are needed to join sub-domains of the mesh. Of less use are the meshes where none of the coordinate lines coincides with the object surface. The basic advantage of such a mesh is that it does not depend on the geometry of the considered object. It should be noted that over the last years the meshes of such a type have got spread by using non-structured meshes. To a certain extent, this approach is universal and does not assume that a mesh is preliminarily set to a given geometry of object. The disadvantages of this approach are
associated with the difficulty to obey the boundary conditions at walls and to obtain an exact solution in their vicinity.

The approaches of component-adaptive overlapping and component-adaptive interface meshes are perspective. The first case deals with two partially overlapped meshes and the second – with joining meshes of different-density computational nodes along the selected interface so that meshes are not overlapped. The first case uses the interpolation at a transition from one mesh to another.

With the procedure of placing mesh nodes, algebraic meshes are the simplest. Such meshes propose that the geometry of sub-domains can be simplified and, accordingly, a less complex procedure of placing computational cells can be used. The disadvantage of such an approach is that it is difficult to control mesh lines in the vicinity of object and inside vortex cells. As a result, a marked slant of mesh lines is seen.

Depending on the type of the considered governing equations, more complex differential equations for mesh are divided into elliptic, hyperbolic, and parabolic. Orthogonal meshes are the most desirable.

By the methods of use of basic elements (border of a mesh and its coordinate system), two alternative numerical methods of discretizing the space are distinguished: feature extraction method and “level set” method (Thompson, 1999). In the first case, the computational domain has the inner boundaries, at which some conditions or relations for physical parameters (shock waves, contact discontinuities, and other front phenomena) must be satisfied and which are defined from the solution. Here the problems arise because the exact position of such boundaries is usually unknown. Because of this, for unsteady-state problems it is required to find the solution in variable-geometry domains. This much complicates the building of universal algorithms and increases the computer resources (Roe, 2001). In the second case, the problems arise because the features of solutions are represented not by the discontinuities of physical parameters, since they are traditionally described in analytical-mathematical models, but by some (usually intermediate) set of values in several discrete elements. As a result, such “discontinuities” are as if smeared out in the space and in some cases they can change fully or even distort the physics of phenomenon and process. However these methods have appeared to be universal and today the algorithms based on them are used everywhere and have become predominant (Vorozhtsov, 1990).

In the course of numerical computations, sometimes it appears to be not only beneficial, but also necessary to vary the parameters of discrete elements. The time history of the mesh space may therefore be an important aspect of problems. So, the mesh
adaptation technique originated from the feature extraction method, in principle, allows one to vary automatically the space discretness degree in different domains without accuracy loss and at that, it can be directly included into the definition of the numerical solution (Liseikin, 1999).

At present, there are many commercial firms that are selling all-kind systems of meshes for any platforms and operational systems. Today, the commercial product most widespread in the world for professional use is represented by ICEM CFD™ (ICEM. Dassault Systemes homepage), as well as by mesh builders most popular among CAD/CAE systems: AutoCAD™, ANSYS™, CATIA™, I-DEAS™, GAMBIT™, etc. Mesh builders permit one not only to build any model, but also to interact with different solvers, as well as to optimize and adapt a mesh according to the features of a particular problem.

The number of meshes reflects the embodiment of one of the first concepts of computer numerical simulation: partition and independent existence of the mathematical model (as a system of differential equations) and the mesh discrete space. The development of numerical methods and the experience of their realization make specific demands. The most substantial demand is associated with the presence of an implicit but close relationship between the geometric parameters of discrete elements of the model and the inner characteristics of the processes to be modeled.

11.2. Some examples of modern CFD software. Basic properties

11.2.1. ANSYS Fluent

ANSYS Fluent is one of the most popular and widespread CFD software in the world. It is composed of several codes used for solving individual applied fluid dynamics problems. Hence, it implies a unique modularity in building a code with a narrow use of separate special modules linked by common GUI (Figure 11.2.1).
ANSYS Fluent uses the generally accepted approach to the choice of special modules. First of all, two flow regimes (steady-state and unsteady-state) and three basic velocity ranges depending on the medium compressibility (incompressible (low subsonic), weakly compressible (trans-sonic) and strongly compressible (super- and hypersonic) flows) are marked out. Laminar, transient, and turbulent flow regimes (in terms of the Reynolds number) and separately Newtonian and non-Newtonian flow regimes (in terms of viscosity) are distinguished. Computations take into consideration all modes of heat and mass transfer, including convection and radiation, mixing of chemical components, chemical reactions, and mass transfer.

ANSYS Fluent computation begins with partitioning the computational domain into the cells that form a mesh, at whose nodes the values of the selected parameters are determined. A mesh can be Cartesian, cylindrical or curvilinear, and surface-adapted. For the coordinate system adapted to the curvilinear surface, a mesh can be extended, compact, rolled or curved in effort to adapt it to a given object geometry. Meshes must be topologically Cartesian, i.e., they must have six faces and eight vertices. As for coordinate lines one should adhere to a certain set of the fundamental rules. No other limitations on the computational mesh are imposed.

After a mesh has been constructed according to design data, a user specifies the fluid properties and boundary conditions. Typical boundary conditions are: assigned mass flowrate, fixed pressure, solid or porous surface, heat intensity, etc.
ANSYS Fluent can solve problems of defining 50 parameters. The physical value of each parameter is defined by a user. Often among these parameters are (FLUENT 6.2 User's Guide):

- pressure;
- three velocity components for each of the available phases;
- enthalpy;
- turbulent characteristics;
- species concentration.

Values of additional parameters such as temperature, density, Mach number or absolute velocity, etc. are determined in terms of those of the above-mentioned parameters. They can be kept at each computational node and stored for a subsequent processing, analysis or graphical representation of the data obtained. This is the content of typical CFD software.

### 11.2.2. Phoenics

Of interest is the evolution of another CFD software called PHOENICS (Parabolic Hyperbolic Or Elliptic Numerical Integration Code Series) (PHOENICS Overview) (Figure 11.2.2), where unlike ANSYS Fluent, the computational algorithm SIMPLE is kept for more than 30 years and is used along with a great number of its modifications supplemented with high-order accuracy finite-difference schemes (QUICK, SMART, B.vanLeer’s et al.).

The CFD software was one of the first software products, starting with which the universal software was under development. The PHOENICS software can be used on many multiprocessor platforms and has extended the capabilities of solvers due to a multigrid MIGAL-algorithm, etc. This complex provides a network access to learning and computing via the Internet.

The time-tested reliability and longevity of the concepts underlying PHOENICS should be emphasized. Up to now, the originally envisaged possibility to compute stresses in a solid body along with flow and heat transfer is absent in other software. This also refers to the multi-fluid model of turbulence (MFM) that is the result of developing the probability density function transport model and the two-phase fluid model. As yet, the long recognized need to allow for turbulent parameter averaging not only on the macro but also
microlevel (combustion, chemistry of mixed components, etc.) is not fully demanded by numerical simulation experience.

Figure 11.2.2. PHOENICS Dialogue window

It should be emphasized that the concept permitting one to assign the objective function for the entire computation is used in the unique constant optimising software package (COSP).

The main idea underlaying PHOENICS, which keeps data formats invariable during a long time is their structuring. Data needed for starting computations are originally partitioned into 24 groups and are located in the so-called Q1-file. This appears to be enough to provide practically all functional capabilities of the software. Of importance is also an additional convention that the data for some not specified group show either their absence or the use of some standard “default” settings. Although most of the CFD softwares use a similar approach, the detailed analysis reveals their substantial differences. The features are associated with the structure of the input data that are defined by that of the software shell that assigns the sequencing to set the software for numerical simulation.
The PHOENICS developers introduced the concept of different-level users (newuser or expert). The software structure permits one on the “expert” level to make alterations caused by the technological development of computers and the advent of theoretical novelties without substantial modifications of the original software code. At that, owing to the modularity and independence of its objects, which is provided by the well thought-out formats of data exchange files between individual modules, the numerical algorithm modifications and the technical plan improvements can, in general, either be hidden to average users or presented at their disposal as a possibility to select some key word “advanced”.

11.2.3. STAR-CD

STAR-CD is the multi-purpose CFD-software (Figure 11.2.3). It was the first software that had included the procedure of “sliding meshes”. Automobile industry was one of the first fields of its wide use. Later STAR-CD began to spread in other areas. The CFD guarantee is its wide cooperation with industrial corporations, engineering firms, and research centers (ADAPCO, Daimler-Chrysler, Imperial College, Aerospatiale-Missiles, etc.).

STAR-CD is closely integrated with such software as ICE-M (Tetra, AutoHexa, MOM3D, Visual3, etc.), imports geometric models in STL, IGES and VDA-FS formats and finite-element models, as well as provides the graphical representation of the results obtained for ANSYS, HEXAR, ICE-M, I-DEAS, NASTRAN, PATRAN, HYPERMESH. Especially in the STAR-HPC parallel version, interfaces were created for ANSYS, NASTRAN, PERMAS, PAM-CRASH, etc. This is associated with developing the coupled fluid-structure interaction technologies (FSI) by joining STAR-CD and different well-known finite-element software.

The created libraries of generic coupling communication interfaces enable one to exchange the mesh topology and the analysis results, as well as to interpolate the obtained results for incompatible meshes by transferring data with a given cell crowding and by setting the next iteration in STAR-HPC and in the finite-element software. This makes it possible to perform a complete joint analysis of dynamic and kinematic, unsteady gasdynamic and non-linear thermal stress computations.
STAR-CD uses very efficient numerical algorithms. As a rule, 40-60 Mb are needed for each 100 000 cells. The STAR-HPC version for supercomputers provides a practically linear increase of the computation rate (57-fold acceleration is attained on the 60 processor-based platform). It is very attractive that apart from computers, the computation parallelization is also supported for distributed memory systems – clusters composed of several computers combined into the network.

Stable numerical procedures provide the possibility to solve large-scale problems on the “expert” level. For example, for flow around a car to be modeled, the Mercedes Benz specialists had to use the model composed of 10mln elements. The solving was made on a 128 processor computer IBM SP2. The storage request was 6Gb. The problem of flow around the car front part and of engine cooling was solved on 64 processors and its dimension was about 7 mln cells. The total solution time of the latter was ~24 hours.

At last, one of the the large-scale problems ever solved in the world – nuclear reactor computation(Star-CD) is the example of the solver power and the parallelization
efficiency. The problem dimension was *250 mln discrete elements and hundreds of millions of freedom degrees*. The solving was made on a cluster comprising 64 IBM SP stations united into the network and took ~55 hours.

The efficient algorithm in STAR-CD is based on the finite-volume method in combination with the SAMM methods for partition of the flow region into discrete elements, which allows one to model problems of any geometry. It should be noted that the SAMM methods are based on the known fractional cell technique (Harlow, 1965).

Owing to this, of interest is the difference in the concepts of CFD-software discretization with the use of the same fractional cell technique but with the space discretization approach different from the SAMM. In the concepts prior to the SAMM it is assumed that the space, which has fallen into the computational domain, can be discretized by a user with no regard to local medium properties and their appropriate space boundaries.

### 11.2.4. Specialized software

Apart from universal complexes for solution of “nonstandard” problems, small research teams are developing many specialized applied softwares. Such softwares are very specific and obey the necessary requirements for accuracy and prediction of physical phenomena of question. They are based on the modern object-oriented programming methods and achievements in the field of numerical computations and algorithms. They gain access to inner codes, which allows a fast renewal in comparison with universal softwares (Быстров, 2005). Therefore the specialized computational softwares reflect modern CFD developments.

**Concluding remarks**

The distinctive features of both universal and specialized CFD softwares are their adaptation to the newest technical capabilities of serial computers by creating the advanced modelling level for “experts”, by structuring data, by selecting solutions in terms of accuracy and computation time, by improving numerical schemes and algorithms with a less computation volume.

In essence, the improvement of discretization methods are aimed at making a compromise between the possible directions and known methods for solution of basic numerical simulation problems. Today, the computation accuracy is to a great extent associated with discrete element sizes and and the accuracy order of numerical schemes.
But its improvement by simply increasing the number of elements is prevented by computer engineering limitations (limited size for data location and code quick-response) that specify the range of phenomena that cannot as yet be solved today.

Here it is quite natural to reduce the number of elements there where the same accuracy can be achieved using larger discrete elements. This is the idea to make computations on nonuniform meshes. Its implementation results in the fact that the increasing algorithmic complexity of methods, as well as the demands for a markedly high level of the user’s special training are of importance.
12. Algorithm of thermo-gas dynamic end heat transfer modeling for turbine blades

Adalat Samadov Soltan, Parviz Abdullayev Shahmurad, Rzagulu Agaverdiyev Sultan

National Academy of Aviation, Aircrafts and Aviation Engines Department, Azerbaijan
12.1. Eulerian models

In Eulerian models the gas and the solid phases are treated as interpenetrating phases and the theory behind such models is basically an extension of the classical kinetic theory that takes non-ideal particle-article collisions and gas-particle drag into account. In this scheme, collections of particles are modelled using continuous medium mechanics. The solid particles are generally considered to be identical having a representative diameter and density, meaning that the particle phase is volume averaged. The general idea in formulating such a multi-fluid model is to treat each phase as an interpenetrating continuum and therefore to construct integral balances of continuity, momentum and energy for both phases with appropriate boundary conditions and jump conditions for the phase interfaces. Since such a resulting continuum approximation for the solid phase has no equation of state and obviously lacks variables such as viscosity and normal stress, certain averaging techniques and assumptions are required to obtain a momentum balance for the solid phase.

Figure 12.1.1 shows a snapshot of liquid fuel spray coming out of an injector nozzle in a realistic gas-turbine combustor. Here the spray atomization was simulated using a stochastic secondary breakup model with point-particle approximation for the droplets. Very close to the injector, it is observed that the spray density is large and the droplets cannot be treated as point-particles. The volume displaced by the liquid in this region is significant and can alter the gas-phase flow and spray evolution.

In order to address this issue, one can compute the dense spray regime by an Eulerian technique using advanced interface tracking/level-set methods. This, however, is computationally intensive and may not be viable in realistic complex configurations. We therefore plan to develop a methodology based on Eulerian-Lagrangian technique which will allow us to capture the essential features of primary atomization using models to capture interactions between the fluid and droplets and which can be directly applied to the standard atomization models used in practice. The numerical scheme for unstructured grids developed for incompressible flows is modified to take into account the droplet volume fraction. The numerical framework is directly applicable to realistic combustor geometries. Our main objectives in this work are:

Develop a numerical formulation based on Eulerian-Lagrangian techniques with models for interaction terms between the fluid and particles to capture the Kelvin-Helmholtz type instabilities observed during primary atomization.

Validate this technique for various two-phase and particulate flows.
Assess its applicability to capture primary atomization of liquid jets in conjunction with secondary atomization models.

![Figure 12.1.1. Snapshot spray from a gas-turbine fuel-injector.](image)

Although constitutive relations according to the kinetic theory of particle flow have been incorporated into recent models, pure CFD models for fluid bed granulation still suffer from the fact that the contact between fluid, particles and boundary surfaces is not considered explicitly with respect to particle inertia and the mechanical properties of the particles. This limits the ability of CFD multiphase models to adequately represent particle-particle and fluid-particle interactions thereby reducing the accuracy of the prediction of both the fluid and the particle dynamics. Considering the required computational power and complexity, gas-particle flow fields calculated with the multi-fluid interpenetrating approach of the Eulerian granular multi-phase model is still a fast method to calculate flow fields, as it is well known from simple particle systems as spray-drying and conveying systems etc. Due to the obvious need for accounting precise particle level properties into fluid bed hydrodynamic models, pure Eulerian CFD models must be regarded as inappropriate even in an industrial context.
12.2. Cooling via internal convective flows

The purpose of cooling technology of Aviation Engines Gas Turbine components via internal convective flows is obtain the highest overall cooling effectiveness with the lowest possible penalty on the thermodynamic cycle performance.

The coolant is extracted from the internal channel for impingement and pin fin cooling. Jet impingement is a very aggressive cooling technique which very effectively removes heat from the vane wall. However, this technique is not readily applied to the narrow trailing edge. The blade trailing edge is cooled using pin-fins (an array of short cylinders).

![Aviation Engine Cross Section](image)

Figure 12.2.1. Aviation Engine Cross Section

The pin-fins increase the heat transfer area while effectively mixing the coolant air to lower the wall temperature of the vanes. After impinging on the walls of the airfoil, the coolant exits the vane and provides a protective film on the blade external surface. Similarly, the coolant travelling through the pin-fin array is ejected from the trailing edge of the airfoil.

The importance of improving the cooling of gas turbine blades have seen an increasing demand of, initially experimental evidences, and now well validated numerical methods for use as optimization tools. The effect of variating rib-sizes, Reynolds number etc. have been documented in both numerical simulations and experimentally.
Although the many merits of experimental evidence, there is a problem that should be recognize when performing experiments: it is very difficult to achieve conditions which enable measured data to be dependent on only a single parameter. In many investigations the data is obscured by slight perturbations in Reynolds number, different heating, rotation number etc. These unknowns add up to a level of uncertainty in the measured data, which should be considered when making comparisons to the predicted result. In numerical simulations on the other hand it is very easy to ensure an exact Reynolds number or that the flow is incompressible, or that the temperature behave as a passive scalar etc., all conditions which can only be an approximation of the real world. It is thus important that both measurements and predictions are made under as identical conditions as possible, to enable an accurate evaluation of turbulence and heat transfer models. The appearance of Direct Numerical Simulations have had a great impact on turbulence modelling as these numerical 'experiments' could be completely controlled, with known boundary conditions.

These simulations also provide a wealth of information. DNS are however very computational costly, and only some fundamental flows can be studied with this approach.
In other cases, such as the internal cooling systems of a gas turbine blade, one needs to rely on data from experiments.

**Impingement Cooling**

Impingement cooling is commonly used near the leading edge of the airfoils, where the heat loads are the greatest. With the cooling jets striking (impinging) the blade wall, the leading edge is well suited for impingement cooling because of the relatively thick blade wall in this area.

![Schematic of a turbine vane cross-section with impingement and trailing edge pin-fin cooling](image)

Figure 12.2.3. Schematic of a turbine vane cross-section with impingement and trailing edge pin-fin cooling

Impingement can also be used near the mid-chord of the vane. Figure 12.2.3 shows jet impingement located throughout the cross-section of an inlet guide vane. Several aspects must be considered when developing efficient cooling designs. The effect of jet-hole size and distribution, cooling channel cross-section, and target surface shape all have
significant effects on the heat transfer coefficient distribution. Jet impingement near the mid-chord of the blade is very similar to impingement on a flat plate; however, the sharp curvature at the leading edge of the vane must be considered when utilizing impingement in this region. As shown in Figure 12.2.3, many jets are used to increase the heat transfer from the vane wall. It has been shown by Taslim and Bethka (2008) that multiple jets perform very differently from a single jet striking a target surface. They concluded that for multiple jets, the Nusselt number is strongly dependent on the Reynolds number, while there is no significant dependence on the jet-to-target plate spacing.

The difference is due to the jet cross-flow from the spent jets. Studies (in Han et al, 2012) by Florschuetz et al., Koopman and Sparrow showed that the mass from one jet moves in the cross-jet flow direction, and this flow can alter the performance of neighbouring jets. The crossflow attempts to deflect a jet away from its impinging location on the target plate. In situations with very strong cross-flow and sufficiently large jet-to-target plate spacing, the cross-flow can completely deflect the jet away from the impingement surface. Florschuetz and Su (in Han et al, 2012) reported that cross-flow decreases the overall heat transfer from the impingement surface. They determined that cross-flow enhances the convective heat transfer, but the enhancement from the jets decreases, as the jets are deflected. Because the enhancement from the impingement jets is much greater than the convective enhancement, the overall Nusselt numbers decrease in the presence of cross-flow.

As shown in Figure 12.2.4 in Ansys program software via internal convective flows throw radial cooling holes obtained the temperature contour. For obtaining temperature distribution in turbine blade 3D model in Ansys 14 software it is necessary to prepare the algorithm of steps as shown below:

1. Step1 – the 3D turbine blade model sketch (2D) preparing in Ansys 14 workbench “geometry” section;
2. Step2 – the 3D turbine blade model preparing in Ansys 14 workbench “geometry” section;
3. Step3 – the 3D turbine blade model mesh generating in Ansys 14 workbench “mesh” section (the same process shown in Fig.12.2.5 in plate);
4. Step4 – in this step blade with generated mesh transferred to in ansys 14 workbench “fluent” section and in this section obtained blade temperature distribution;
5. Step5 – for obtaining the results of temperature distribution in blade the solution transferred to Ansys 14 workbench “result” section (the temperature contour obtained in mentioned section shown in Fig.12.2.4).

![Temperature Contour](image)

**Figure 12.2.4. Temperature contour**

In Figure 12.2.5 you can see the plate with rectangular mesh. A typical test model is shown in Figure 12.2.6. As shown in this figure, the coolant jets impinge on the target surface from the jet plate in an inline array (Kwiatkowski and Domanski, 2012). As the coolant travels along the test surface, the spent air from the upstream jets effects the heat transfer coefficient distributions of the downstream jets, and this effect increases as more spent air accumulates on the target surface. Obtained temperature contour shown in Figure 12.2.6.

Correlations based on experimental data were developed by Kercher and Tabakoff, and Florschuetz et al. (Kercher and Tabakoff, 2012) to estimate the heat transfer enhancement from an array of impinging jets. Although the correlations are in different forms, they both demonstrate the dependence of the heat transfer enhancement on the amount of cross-flow. Florschuetz et al. (Kercher and Tabakoff, 2012) also showed the cross-flow effect is
much stronger in staggered arrays of jets than an inline array. Bailey and Bunker extended the correlation developed by Florschuetz et al. (Kercher and Tabakoff, 2012) to include the effect of jet spacing. The correlation has been extended to include dense impingement arrays.

Figure 12.2.5. Plate (with rectangular mesh)

By mentioned method controlled the direction of the cross-flow and obtained detailed distributions of the heat transfer coefficients for three target plates. The results clearly indicate when the cross-flow travels in two opposite directions; the heat transfer enhancement on the target plate is much greater than when the cross-flow is restricted to one direction. The heat transfer enhancement on the target plate decreases near the edges due to the decreased coolant flow (for film cooling). This study also concluded that increasing cross-flow results in degraded heat transfer; however, the heat transfer coefficient distribution is much more uniform.

The heat transfer coefficient distributions on target plates with stretched arrays of impinging jets were by cross-flow method.
This array varies from the traditional square array in which the jets are evenly spaced. They concluded the existing correlations for square arrays over-predict the effect of cross-flow in the target surface. The presence of initial cross-flow also affects the heat transfer enhancement from the target plate. The cross-flow described above is created by the spent flow from the jets. Therefore, the first row of jets is not affected by the cross-flow. However, in many situations, cross-flow may develop upstream of the first row. The flow from upstream of the impingement jets can significantly alter the flow near the jets, and thus alter the heat transfer coefficients on the target surface. Florschuetz et al. (Kercher and Tabakoff, 2012) investigated the effect of initial cross-flow on the heat transfer enhancement. The results of this study were similar to those mentioned above describing cross-flow: the heat transfer enhancement on the target plate decreases when initial cross-flow is present.

Figure 12.2.6. The coolant jets impinge on the target surface from the jet plate

One of the main directions for the performance efficiency improvement of the power system in the modern aviation engines include increasing of the work flow conditions, and most of all it’s come to the temperature increasing in the gas turbine inlet. In this case the most complicated problem is the design reliability assurance of the nozzle and rotating blades of the gas turbine, which falls under the direct effect of the high potential gas flow and significant loads. Therefore improvement and defining the ways for thermal protection, the lifetime and reliability increasing of the turbine details have a great scientific and practical importance.
The activities for the temperature increasing can be compared to several directions. However assimilation of high $T_g$ in AGTE mainly reached by refinement of the cooling systems for turbine blades. It is especially necessary to note, that with $T_g$ increase the requirement to accuracy of results will increase. In other words, at allowed values of AGTE metal temperature $T_{lim} = (1100...1300K)$, the absolute error of temperature calculation should be in limits $(20 – 30K)$, that is no more than 2-3%. This is difficult to achieve (multi-connected fields with various cooling channels, variables in time and coordinates boundary conditions). Such problem solving requires application of modern and perfect mathematical device.

In classical statement a heat conduction differential equation in common case for non-stationary process with heat distribution in multi-dimensional area (Fourier-Kirchhoff equation) has a kind (Leontiev et al., 2004), (Zisina-Molojen et al., 1974):

$$\frac{\partial(\rho C_v T)}{\partial t} = \text{div}(\lambda \text{grad} T) + q_v,$$

where $\rho$, $C_v$, and $\lambda$ – accordingly material density, thermal capacity, and heat conduction; $q_v$ – internal source or drain of heat; and $T$ – required temperature.

Research has established that the temperature condition of the airfoil with radial cooling channels can be determined as two-dimensional (Leontiev et al., 2004), (Zisina-Molojen et al., 1974). Besides, if to suppose constancy of physical properties and absence of internal sources (drains) of heat, then the temperature field under fixed conditions will depend only on the skew shape and on the temperature distribution on the skew boundaries. In this case, equation (12.1) will look like:

$$\Delta T = \frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} = 0$$

When determining particular temperature fields in gas turbine elements are used boundary conditions of the third kind, describing heat exchange between the skew field and the environment (on the basis of a hypothesis of a Newton-Riemann). In that case, these boundary conditions will be recorded as follows:

$$\alpha_0(T_0 - T_{\gamma_0}) = \lambda \frac{\partial T_{\gamma_0}}{\partial n}$$
This following equation characterizes the quantity of heat transmitted by convection from gas to unit of a surface of a blade and assigned by heat conduction in a skew field of a blade.

\[- \lambda \frac{\partial T_{y_i}}{\partial n} = \alpha_i (T_{\gamma_i} - T_i) \quad (12.4)\]

Equation (12.4) characterizes the heat quantity assigned by convection of the cooler, which is transmitted by heat conduction of the blade material to the surface of cooling channels: where \( T_0 \) – temperature of environment at \( i = 0 \); \( T_i \) – field temperature at \( i = I, M \) (temperature of the cooler), where \( M \) – quantity of outlines; \( T_{\gamma_0} \) – temperature on an outline \( \gamma_i \) at \( i = 0 \) (outside outline of blade); \( T_{\gamma_i} \) – temperature on an \( \gamma_i \) at \( i = I, M \) (outline of cooling channels); \( \alpha_0 \) – heat transfer factor from gas to a surface of a blade (at \( i = 0 \)); \( \alpha_i \) – heat transfer factor from a blade to the cooling air at \( i = I, M \); \( \lambda \) – thermal conductivity of the material of a blade; \( n \) – external normal on an outline of researched area.

For the solution of this boundary problem (12.2)-(12.4) four numerical methods are used: Methods of Finite Differences (MFD), Finite Element Method (FEM), probabilistic method (Monte-Carlo method), and Boundary Integral Equations Method (BIEM) (or its discrete analog — Boundary Element Method (BEM)).

Let us consider BIEM application for the solution of problem (12.2)-(12.4) (Leontiev et al., 2004), (Zisina-Molojen et al., 1974), (Pashayev et al., 2005).

The function \( T = T(\xi, y) \), continuous with the derivatives up to the second order, satisfying the Laplace equation in considered area, including and its outline \( \Gamma = \bigcup_{i=0}^{M} \gamma_i \), is harmonic. Consequence of the Grin integral formula for the researched harmonic function \( T = T(\xi, y) \) is the ratio:

\[
T(x, y) = \frac{1}{2\pi} \int_{\Gamma} \left[ T_{\Gamma} \frac{\partial (\ell n R)}{\partial n} - \ell n R \frac{\partial T_{\ell}}{\partial n} \right] ds
\quad (12.5)
\]

where \( R \) – variable at an integration of the distance between point \( K(\xi, y) \) and “running” on the outline \( k \)- point; \( T_{\Gamma} \) – temperature on the outline \( \Gamma \). The temperature value in
some point \( k \) lying on the boundary is determined (as limiting at approach of point \( K(\xi,\gamma) \) to the boundary):

\[
T_k = \frac{1}{2\pi} \left[ \int_{\Gamma} T \frac{\partial (\ell n R_k)}{\partial n} ds - \int_{\Gamma} \frac{\partial T}{\partial n} \ell n R_k ds \right]
\]  

(12.6)

With allowance of the boundary conditions (12.2)-(12.3), after collecting terms of terms and input of new factors, the ratio (12.6) can be presented as a linear algebraic equation, computed for the point \( R \):

\[
\varphi_{k1} T_{\gamma_{01}} + \varphi_{k2} T_{\gamma_{02}} + \cdots + \varphi_{kn} T_{\gamma_{0n}} - \varphi_{k\gamma_0} T_0 - \varphi_{k\gamma_i} T_i - 2\pi T_k = 0
\]

(12.7)

where, \( n \) is the quantity of sites of a partition of an outside outline of a blade \( \ell_\gamma (\ell_\gamma \text{ on } i = 0) \) on small sections \( \Delta S_0 (\Delta S_i \text{ at } i = 0) \), \( m \) is the quantity of sites of a partition of outside outlines of all cooling channels \( \ell_\gamma (i = 1, M) \) on small sections \( \Delta S_i \).

Let us note, that unknowns in the equation (12.7) except the unknown of true value \( T_k \) in the \( k \) point are also mean on sections of the outlines partition \( \Delta S_0 \) and \( \Delta S_i \) temperatures \( T_{\gamma_{01}}, T_{\gamma_{02}}, \ldots, T_{\gamma_{0n}} \) and \( T_{\gamma_{i1}}, T_{\gamma_{i2}}, \ldots, T_{\gamma_{im}} \) (total number \( n + m \)).

From a ratio (12.7), we shall receive the required temperature for any point, using the formula (12.5):

\[
T(x,y) = \frac{1}{2\pi} \left[ \varphi_{k1} T_{\gamma_{01}} + \varphi_{k2} T_{\gamma_{02}} + \cdots + \varphi_{kn} T_{\gamma_{0n}} + \cdots + \varphi_{kn} T_{\gamma_{im}} - \varphi_{k\gamma_0} T_0 - \varphi_{k\gamma_i} T_i \right]
\]

(12.8)

where

\[
\varphi_{hi} = \frac{\alpha_{hi}}{\lambda_i} \int_{\Delta S_i} \frac{\partial (\ell n R_{hi})}{\partial n} ds - \frac{\alpha_{hi}}{\lambda_i} \int_{\Delta S_i} \ell n R_{hi} ds
\]

...
In activities (Zisina-Molojen et al., 1974) the discretization of a line \( \Gamma = \bigcup_{i=0}^{M} \gamma_i \) by a many discrete point and integrals that are included in the equations as logarithmic potentials, was calculated approximately with the following ratios:

\[
\int_{\Delta S_{\gamma_i}} \frac{\partial (\ell n R_k)}{\partial n} ds \approx \frac{\partial (\ell n R_k)}{\partial n} \Delta S_{\gamma_i},
\]

(12.9)

\[
\int_{\Delta S_{\gamma_i}} \ell n R_k ds \approx \ell n R_k \Delta S_{\gamma_i},
\]

(12.10)

where \( \Delta S_{\gamma_i} \in L = \bigcup_{i=0}^{M} 1_i \); \( 1_i = \int_{\gamma_i} ds \).

In contrast to (Zisina-Molojen et al., 1974), we offer to decide the given boundary value problem \((12.2)-(12.4)\) as follows. We locate the distribution of temperature \( T = T(x, y) \) as follows:

\[
T(x, y) = \int_{\Gamma} \rho (\ell n R^{-1}) ds,
\]

(12.11)

where \( \Gamma = \bigcup_{i=0}^{M} \gamma_i \) - smooth closed Jordan curve; \( M \) - quantity of cooled channels; \( \rho = \bigcup_{i=0}^{M} \rho_i \) - density of a logarithmic potential uniformly distributed on \( \gamma_i S = \bigcup_{i=0}^{M} S_i \).

Thus curve \( \Gamma = \bigcup_{i=0}^{M} \gamma_i \) are positively oriented and are given in a parametric kind: \( x = x(\xi); y = y(\xi); s \in [0, L]; L = \int_{\Gamma} ds \).

Using BIEM and expression (12.11) we shall put problem \((12.2)-(12.4)\) to the following system of boundary integral equations:

\[
(\rho(s) - \frac{1}{2\pi}) \int_{\Gamma} (\rho(s) - \rho(\xi)) \frac{\partial}{\partial n} \ell n R(s, \xi)d\xi = \frac{\alpha_s}{2\pi\lambda} (T - \int_{\Gamma} \rho(s)/\ell n R^{-1} ds),
\]

(12.12)

where \( R(s, \xi) = ((x(s) - x(\xi))^2 + (y(s) - y(\xi))^2)^{1/2} \).

For the singular integral operator’s evaluation, which are included in (12.12) the discrete operators of the logarithmic potential with simple and double layer are
investigated. Their connection and the evaluations in modules term of the continuity (evaluation such as assessments by A. Zigmound are obtained) is shown

Theorem (main):

\[
\int_0^\infty \frac{\omega_j(x)}{x} \, dx < +\infty
\]  
(12.13)

And let the equation (12.12) have the solution \( f^* \in C_\Gamma \) (the set of continuous functions on \( \Gamma \)). Then \( \exists N_0 \in \mathbb{N} = 1, 2, \ldots \) such that the discrete system \( \forall N > N_0 \), obtained from (12) by using the discrete double layer potential operator (its properties has been studied), has unique solution \( \{ \tilde{f}_j^{(N)} \}, j = \overline{1, n} \):

\[
|f^*_j - \tilde{f}_j^{(N)}| \leq C(\Gamma) \left( \int_0^{\varepsilon} \frac{\omega(x) \omega_j(x)}{x} \, dx + \varepsilon \int_{\varepsilon}^{\infty} \frac{\omega(x) \omega_j(x)}{x} \, dx + \omega_j(\|f\|) \right) \left( \int_0^{\infty} \frac{\omega_j(x)}{x} \, dx \right) \]  
(12.14)

where \( C(\Gamma) \) is constant, depending only on \( \|f_N\|_{N=1}^{\infty} \) -- the sequence of partitions of \( \Gamma \); \( \{ \varepsilon_N \}_{N=1}^{\infty} \) -- the sequence of positive numbers such that the pair \( (\|f_N\|, \varepsilon_N) \), satisfies the condition \( 2 \leq \varepsilon \|f\| \leq p \).

Let \( \delta \in \mathbb{Q}, d/2 \), where \( d \) is diameter \( \Gamma \), and the splitting \( \tau \) is that, which is satisfied the condition

\[
p' \geq \delta/\|f\| \geq 2
\]  
(12.15)

Then for all \( \psi \in C_\Gamma \) (\( C_\Gamma \) - space of all functions continuous on \( \Gamma \)) and \( z \in \Gamma \), \( z = x + iy \)

\[
\left| \left( \psi \right|_{\varepsilon, \delta} f(z) - \tilde{f}(z) \right| \leq C(\Gamma)
\]  
(12.16)

\[
\left| \left( \omega_{\tau, \varepsilon} f(z) \right|_{\Omega, \tau} - \tilde{f}(z) \right| \leq C(\Gamma) \left( \int_0^\infty \frac{\omega_j(x) \omega_j(x)}{x^2} \, dx + \omega_j(\|f\|) \int_{\varepsilon}^{\infty} \frac{\omega_j(x)}{x^2} \, dx \right)
\]  
(12.17)
where

\[ \zeta(\tau, \delta) = \sum_{\tau, \delta} \left( \frac{f(z_{k,e+1}) + f(z_{k,e})}{2} - f(z) \right) (y_{k,e+1} - y_{k,e})(x_{k,e} - x) - (x_{k,e+1} - x_{k,e})(y_{k,e} - y) \]

\[ \frac{1}{|z - z_{k,e}|^2} + \mathfrak{f}(z) \]

\( \zeta(\tau, \delta) \) - two-parameter quadrature formula (depending on \( \tau \) and \( \delta \) parameters) for logarithmic double layer potential; \( \mathfrak{f}(z) \) - double layer logarithmic potential operator; \( C(\Gamma) \) – constant, dependent only from a curve \( \Gamma \); \( \omega_f \) is a module of a continuity of functions \( f \):

\[ \zeta(\tau, \delta) = \sum_{\tau, \delta} \frac{f(z_{k,i+1}) + f(z_{k,i})}{2} \ln \frac{1}{|z_{k,i+1} - z_{k,i}|} \] (12.18)

\( \zeta(\tau, \delta) \) - two-parameter quadrature formula (depending on \( \tau \) and \( \delta \) parameters) for logarithmic potential simple layer; \( \mathfrak{f}(z) \) – simple layer logarithmic potential operator;

\[ z_{k,e} \in \tau; \ z_{k,e} = x_{k,e} + iy_{k,e}; \tau(z) = \mathfrak{f}(z) = \sum_{k=1}^{m} \left| z_{k,e} - z \right| > \varepsilon; \] \( \emptyset_{k} = \{ z_{k,1}, \ldots, z_{k,m_k} \} \),

\[ z_{k,1} \leq z_{k,2} \leq \ldots \leq z_{k,m_k}; \| \tau \| = \max_{j=1,m_k} |z_{k,j+1} - z_{k,j}| \] (12.19)

Thus are developed effective from the point of view of realization on computers the numerical methods basing on constructed two-parametric quadrature processes for the discrete operators logarithmic potential of the double and simple layer. Their systematic errors are estimated; the methods quadratures mathematically are proved for the approximate solution Fredholm I and II boundary integral equations using Tikhonov regularization and are proved by appropriate theorems (Pashayev et al., 2002, 2005).

The given calculating technique of the blade temperature field can be applied also to blades with the plug-in deflector. On consideration blades with deflectors in addition to boundary condition of the III kind adjoin also interfaces conditions between segments of the outline partition as equalities of temperatures and heat flows

\[ T_v(x, y) = T_{v+1}(x, y) \] (12.20)

\[ \frac{\partial T_v(x, y)}{\partial n} = \frac{\partial T_{v+1}(x, y)}{\partial n} \] (12.21)
where \( \nu \) – number of segments of the outline partition of the blade cross-section; \( x, y \) – coordinates of segments. At finding of cooler \( T \) best values, is necessary to solve the inverse problem of heat conduction. For it is necessary at first to find solution of the heat conduction direct problem with boundary condition of the III kind from a gas leg and boundary conditions I kinds from a cooling air leg

\[
T_v(x, y)\big|_{y_0} = T_{i0}
\]  

(12.22)

where \( T_{i0} \) – the unknown optimum temperature of a wall of a blade from a leg of a cooling air.

The developed technique for the numerical solution of stationary task of the heat conduction in cooled blades can be distributed also to quasistationary case.

Let us consider a third boundary-value problem for the heat conduction quasilines equation:

\[
\frac{\partial}{\partial x}\left( \lambda(T) \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial y}\left( \lambda(T) \frac{\partial T}{\partial y} \right) = 0
\]  

(12.23)

\[
\alpha_i(T_{ci} - T_{c}) - \lambda(T) \frac{\partial T_{c}}{\partial n} = 0
\]  

(12.24)

For linearization of tasks (12.23) - (12.24) we shall use the Kirchhoff permutation:

\[
A = \int_0^T \lambda(\xi)d\xi
\]  

(12.25)

Then equation (12.23) is transformed into the following Laplace equation:

\[
\frac{\partial^2 A}{\partial x^2} + \frac{\partial^2 A}{\partial y^2} = 0
\]  

(12.26)

For preserving convection additives in boundary-value condition (12.24), we shall accept in initial approximation \( \lambda(T) = \lambda_c \). Then from (12.25) we have

\[
T = A / \lambda_c.
\]  

(12.27)

And the regional condition (12.24) will be transformed as follows:
\[
\alpha_i (T_{ci} - A_{ji} / \lambda_c) - \frac{\partial A_{ji}}{\partial n} = 0
\] (12.28)

So, the stationary problem (12.26) with (12.28) is solved by boundary integrated equations method.

The most important role in the blade raw analysis and improvement belongs to the aerodynamic estimation methods, which proceed to the gas-dynamic efficiency estimation and for the heat exchange problem research.

On the basis of potential flow theory, the velocity distribution of gas flow around the airfoil has been defined by solving the following integral equations for the complex potential flow \( W \) (Pashayev et al., 2004) (alternate to existing (Beknev et al., 1997) by effective realization):

\[
W \equiv \varphi \text{ \bigwedge } i \psi \text{ \bigwedge }
\] (12.29)

integral equations for the velocity potential:

\[
\varphi (x_k, y_k) = V_\infty \text{ \bigwedge } x_k \cos \alpha_\infty + y_k \sin \alpha_\infty - \frac{1}{2\pi} \Gamma \theta_{out} \mp \frac{1}{2\pi} \oint_{S_{out}} \varphi \text{ d}\theta
\] (12.30)

and integral equation for the stream function \( \psi \):

\[
\psi = V_\infty \cos \alpha_\infty - x \sin \alpha_\infty - \frac{1}{2\pi} \oint_{S_{out}} \text{ ln } \sqrt{\frac{sh^2 + x^2 + y^2}{x^2 + y^2}} \text{ d}s
\] (12.31)

which based on the theory of the zero stream function on airfoil contour.

In this: \( \varphi (x_k, y_k) \) – the value of velocity potential; \( V_\infty \) – the main flow velocity; \( \alpha_\infty \) – the angle between the velocity vector \( V_\infty \) and the blade raw axis; \( \Gamma \) – the velocity circulation; \( \theta_{out} \) – the angle that corresponds to the outlet edge of the airfoil.

The integral equation (12.30) solved by the following received approximating equation:

\[
\varphi_j \mp \sum_{i=1}^{n} \varphi_i \text{ \bigwedge } \theta_{j,i-1} \mp V_\infty \text{ \bigwedge } x_k \cos \alpha_\infty + y_k \sin \alpha_\infty - \frac{1}{2\pi} \Gamma \theta_{j,i} \text{ d}s
\] (12.32)

where, \( i = 2n - 1 \), \( j = 2n \), \( n \) is the number of divided parts of the airfoil.
The velocity potential $\phi$ distribution along the airfoil has been received by solving the system of the linear algebraic equations. The flow velocity magnitude determined using velocity potential differentiation along the airfoil contour $s$ , i.e. $V \equiv d\phi/ds$.

As opposite to potential flow, the heat-conducted viscous turbulent flow in the turbine blade channel is explored based on Navier-Stokes equations. As a turbulence models the standard $k - \varepsilon$ (Launer and Spalding, 1974) and Spalart-Allmaras models used (Spalart and Allmaras, 1992). The Navier-Stokes equations solved by ANSYS CFX10© (based on finite element method) and Fluent© (based on control volume method) software’s. As the meshing tool Gambit© software, and for 3D modeling – Solid Works10© have been used.

The velocity distribution data along the airfoil are incoming for determining outer boundary heat exchange conditions. In case of potential flow, the heat transfer coefficient of gas flow defined by using integral equation for the energy of heat boundary layer (Zisina-Molojen, 1974), (Galitseiskiy et al., 1996), (Kopelev and Slitenko, 1994). Correction to the base values of heat coefficient was made by using the recommendations that was approved by experiment-calculated way (Zisina-Molojen, 1974), (Galitseiskiy, 1996), (Kopelev and Slitenko, 1994). For the internal heat exchange boundary conditions determination the correlation between inner geometry and hydrodynamic with heat models are used.

At known geometry of the cooling scheme, for the convective heat exchange local coefficients $\alpha_e$ definition of the cooler by the standard empirical formulas it is necessary to have the basic data’s of the air flow distribution in cooling channels.

For example, for blades with deflector and with cross current, the value of the airflow $G_c$ for blade cooling can be defined with the following:

$$G_c = \frac{\mu_c F_c}{d_c} \left( \frac{d_c}{\lambda_c \cdot C} \right)^{\frac{1}{n}} \left[ \alpha_g \left( \frac{\gamma_g - 1}{\kappa_f} \right) \frac{\gamma_g - 1}{\kappa_f + 1} - \psi_g \right]^{\frac{1}{n}}$$

(12.33)

where $\psi_g$, $\psi_c$ – the gas and air temperature coefficients; $\kappa_f$ – coefficient of the form; $d_c$ – the characteristic size in the formula $\text{Re}_c$; $\mu_c$, $\lambda_c$ – cooler dynamic viscosity and heat conductivity coefficients; $Bi$ – the Bio criterion for the blade wall; $F_c$ – the total area of
passage for air; $C$ and $n$ – coefficient and exponent ratio in criteria formulas for convective heat exchange $Nu_c = CRe^nu_c$ for considered cooling parts.

To determine the distribution of flow in the blade cooling system, an equivalent hydraulic scheme is built.

The construction of the equivalent hydraulic tract circuit of the vane cooling is connected with the description of the cooled vane design. The whole passage of coolant flow is divided in some definite interconnected sections, the so-called typical elements, and every one has the possibility of identical definition of hydraulic resistance. The points of connection of typical elements are changed by node points, in which the streams, mergion or division of cooler flows is taking places proposal without pressure change. All the typical elements and node points are connected in the same sequence and order as the tract sites of the cooled vane.

To describe the coolant flow at every inner node the 1st low by Kirchhoff is used (Galitseiskiy et al., 1996), (Kopelev and Slitenko, 1994), (Arseniev et al., 1985):

$$f_1 = \sum_{j=1}^{m} G_{ij} = \sum_{j=1}^{m} \text{sign} \left( p_{ij} k_{ij} \sqrt{\Delta p_{ij}} \right); \quad i = 1, 2, 3...n \quad (12.34)$$

where $G_{ij}$ – discharge of coolant on the element; $i – j$, $m$ – number of typical elements connected to $i$ node of the circuit; $n$ – number of inner nodes of hydraulic circuit; $\Delta p_{ij}$ – losses of total pressure of the coolant on element $i – j$. In this formula the coefficient of hydraulic conductivity of the circuit element ($i – j$) is defined as:

$$k_{ij} = \sqrt{2 \frac{f_{ij}^2 \cdot p_{ij}}{\xi_{ij}}} \quad (12.35)$$

where $f_{ij}, p_{ij}, \xi_{ij}$ are the mean area of the cross-section passage of elements ($i – j$), density of coolant flow in the element, and coefficient of hydraulic resistance of this element. The system of nonlinear algebraic equations (23) is solved by the Zeidel method with acceleration, taken from:

$$p_i^{k+1} = p_i^k - f_i^k / \frac{\partial f}{\partial p} \quad (12.36)$$

where $k$ – iteration number; $p_i^k$ – coolant pressure in $i$ node of the hydraulic circuit. The coefficients of hydraulic resistance $\xi_{ij}$ used in (12.35) are defined by analytical
dependencies, which are in the literature available at present (Kopelev and Slitenko, 1994), (Arseniev et al., 1985).

For example, to calculate a part of the cooling tract that includes the area of deflector perforation coefficients of hydraulic resistance in spray (Arseniev et al., 1985):

$$\xi_{c_{\Sigma}} = \frac{G_r / G_k^{m}}{2f_c / f_k^{0.1}}, \quad m = \frac{1}{2f_c / f_k^{0.23}}$$

(12.37)

and in general channel:

$$\xi_{c_{\Sigma}} = 1.1G_r / G_k^{m}, \quad n = 0.381\ln f_c / f_k^{0.3}$$

(12.38)

In these formulas, $G_r, G_k$ are cooling air consumption in the spray stream through the perforation deflector holes and slot channel between the deflector and vanes wall, and $f_r, f_k$ - the flow areas.

For the verification of described methods the computational data’s of the first stage nozzle blades of high pressure turbine are used. The nozzle array of test experiment was analytically formed for transonic speed. The blade profile has the following geometrical parameters: blade chord $b = 60\text{mm}$; array spacing $t = 41.5\text{mm}$; throat section width $a_2 = 11.26\text{mm}$; the exit edge depth $d_2 = 1.8\text{mm}$; bend angle of the exit edge $\delta = 12^\circ 48^\prime$. The computing experiment based on the following gas flow parameters: blade raw inlet velocity $V_1 = 156\text{m/s}$; outlet velocity $V_2 = 512\text{m/s}$; outlet angle $\alpha_2 = 16^\circ 6^\prime$; inlet stagnation temperature and pressure $T_g^* = 1193\text{K}, \quad p_g^* = 1.2095\text{MPa}$; outlet conditions $T_{z_1} = 1142\text{K}, \quad p_{z_1} = 0.75\text{MPa}$; relative outlet velocity $\lambda_{j,ad} = 0.891$.

The Figure (12.2.7) represents the velocity distribution along the airfoil. In case of potential flow the results was obtained by the solution of boundary integral equation (\(\bullet\)) (Pashayev et al., 2004). On the same figure the Navier-Stokes differential equations system’s solution is displayed (\(\blacksquare\)), which obtained by ANSYS for the viscous turbulent flow using the $k-\varepsilon$ turbulence model.

On the Figure (12.2.8) the computational domain represents, which received by the Gambit meshing tool. The 2D gas flow with the $k-\varepsilon$ turbulence model is displayed on Figure (12.2.9). The Figure (12.2.10) describes the velocity distribution along the airfoil
for the same flow conditions by using of the Spalart-Allmaras turbulence model with Fluent solver.

Figure 12.2.7. Velocity distribution along the airfoil: ♦ - solution of boundary integral equation for the potential flow; ■ - solution of Navier-Stokes differential equations system; $V$ - gas velocity, m/s; $S/l$ - relative length.

Figure 12.2.8. Computation domain
Figure (12.2.11) and (12.2.12) shows the temperature distribution of the gas flow in the blade channel, which determined with $k-\varepsilon$ and Spalart-Allmaras turbulence model accordingly.

Figure 12.2.9. 2D viscous compressible gas flow with $k-\varepsilon$ turbulence model by ANSYS

Figure 12.2.10. Velocity distribution along the airfoil (m/s), Spalart-Allmaras turbulence model, Fluent

Figure 12.2.11. Static temperature field around the airfoil, Ansys, $k-\varepsilon$ turbulence model

Figure 12.2.12. Static temperature field around the airfoil (K), Fluent, Spalart-Allmaras
By using these results the cooling tract equivalent scheme is developed (Figure 12.2.13) and temperature field of blade cross section (Figure 12.2.14) are determined.

![Figure 12.2.13. The equivalent hydraulic scheme of experimental nozzle](image)

![Figure 12.2.14. Distribution of temperature along outside (▲) and internal (■) contours of the cooled nozzle blade](image)

The calculations reproduce good flow structure as well as local characteristics of the velocity and temperature distribution. Received results approve the accuracy of described methods and their validity the temperature field researches in gas-turbine engines.

On basis of acquired calculations the 3D geometrical model of the cooled nozzle blade is developed, and expressed on Figure (12.2.15).

The multiples computing experiments with the using BIEM for calculation the temperature fields of nozzle and working blades with various amount and disposition of cooling channels, having a complex configuration, is showed, that for practical calculations in this approach, offered by us, the discretization of the integrations areas can be conducted with smaller quantity of discrete points. Thus the reactivity of the algorithms developed and accuracy of evaluations is increased. The accuracy of temperatures
calculation, required consumption of the cooling air, heat flows, and losses from cooling margins essentially depends on reliability of boundary conditions, included in calculation of heat exchange.

Figure 12.2.15. 3D model of the 1st stage nozzle blade of the high pressure gas-turbine

The reliability of the methods was proved by experimental investigations for the heat and hydraulic characteristics of blades. Geometric model, equivalent hydraulic schemes of cooling tracks have been obtained, cooler parameters and temperature field of gas 1st stage turbine nozzle blade of the have been determined. Methods have demonstrated high efficiency at repeated and polivariant calculations on the basis of which was offered as the way of blade cooling system modernization.

12.3. Fundamentals of film cooling performance

For engineers the biggest problem is to design an effective cooling system to protect turbine materials from degradation. For this purpose many modern numerical programs
are commonly used. However, these tools require good solver configurations, well-prepared geometries with meshes and proper boundary conditions. In many cases like for, example, turbine blade cooling, it is hard to model an entire machine because of the limited computational power of commonly used computers. We need accurate methods with reasonable system requirements. To this end engineers analyse only parts of machines, especially when the geometry is periodic (such as in a turbine). Calculations for an entire row of blades may be very time consuming, whereas modelling a single blade with proper boundary conditions and solver settings should still deliver accurate results. The aim of this work is to determine the temperature distribution in the exemplary turbine blade with internal cooling only. Regions of additional cooling requirements in such case will be defined. The geometry and 3D model of the turbine blade was prepared in Unigraphics NX 7.0 using an airfoil contour extracted from the sketch of US patent no. 4753575 (Figure 12.3.1) (Bunker, 2006). The obtained airfoil was different from the original one due to the low quality of the sketch and the approximation done in the digitizer used to obtain coordinates of points on the airfoil. These points were used to create a sketch in NX which, after extruding, gave the outer surfaces of the blade. The internal cooling passages were subsequently created. The radial and fan shaped cooling holes of the turbine blade created with high accuracy.

This modelled system consisted of internal cooling and film cooling technology. At the trailing edge of blade created turbulator ribs for effective cooling of trailing edge. That is the main reason why in mentioned construction has 2 types of cooling holes: radial and fan shaped cooling holes.

At the next step for 3D model generating, heat exchanger and cooling solutions geometry was exported as a Parasolid file and imported to Ansys 14 program software (Figure 12.3.2) which combined with Fluent program software. In Ansys 14 performed mesh generating, temperature solution and distribution along the height of turbine blade (Figure 12.3.3). In fluent is chosen Spalart-Allmaras model as viscosity model.
For generating 3D model in ansys 14 software it is necessary to prepare the algorithm of steps as shown below:

1. Step1 – the 3D turbine blade model sketch (2D) preparing in Ansys 14 workbench “geometry” section;
2. Step2 – the 3D turbine blade model preparing in Ansys 14 workbench “geometry” section (as shown Fig.12.3.2);
3. Step3 – the 3D turbine blade model mesh generating in ansys 14 workbench “mesh” section;
4. Step4 – In this step blade with generated mesh transferred to in Ansys 14 workbench “fluent” section and in this section obtained blade temperature distribution;
5. Step5 – For obtaining the results of temperature distribution in blade the solution transferred to ansys 14 workbench “result” section (the temperature contour obtained in mentioned section shown in Fig.12.3.3).

The primary process by which film cooling reduces the heat transfer to the wall is by reducing the gas temperature near the wall, i.e. reducing the driving temperature potential for heat transfer to the wall.

![3D model of turbine blade](image)

**Figure 12.3.2.** 3D model of turbine blade

As the coolant flows from the coolant holes, it mixes with the mainstream gas resulting in an increase in coolant temperature. A typical example of this is presented in Figure 12.3.3 which shows measurements of the temperature profile along the centerline of a coolant jet as it flows downstream of the coolant holes.

In this figure presented the temperature contour of turbine blade which generating via Ansys and Fluent software. As the turbine inlet temperature increases, the heat transferred to the turbine blade also increases. The level and variation in the temperature within the blade material, which cause thermal stresses, must be limited to achieve reasonable durability goals. The operating temperatures are far above the permissible metal temperatures. Therefore, there is a critical need to cool the blades for safe operation. The blades are cooled with extracted air from the compressor of the engine. Since this extraction incurs a penalty on the thermal efficiency and power output of the engine, it is important to understand and optimize the cooling technology for a given turbine blade geometry under engine operating conditions.

Gas turbine cooling technology is complex and varies between engine manufacturers. Figure 12.3.4 shows the common cooling technology with three major internal cooling zones in a turbine blade with strategic film cooling in the leading edge, pressure and
suction surfaces, and blade tip region. The leading edge is cooled by jet impingement with film cooling, the middle portion is cooled by serpentine rib-roughened passages with local film cooling, and the trailing edge is cooled by pin fins with trailing edge injection. This article focuses on the rotational effects on the turbine blade internal cooling passage heat transfer and the unsteady high free-stream turbulence effects on the turbine blade film cooling performance with standard and shaped film-hole geometry.

![Temperature contour of turbine blade](image)

**Figure 12.3.3. Temperature contour of turbine blade**

Interested readers are referred to several recent publications that address state-of-the-art reviews of turbine blade cooling and heat transfer.

These include rotational effect on the turbine blade coolant passage heat transfer by Dutta and Han (2012), recent developments in turbine blade film cooling and recent developments in turbine blade internal cooling by Han et al. (2012). A recent book
focusing entirely on the range of gas turbine heat transfer issues and the associated cooling technology is available by Han et al. (2012), Taslim and Bethka (2008). A symposium volume dealt with heat transfer in gas turbine systems is recently edited by Goldstein (Han et al., 2012). A detailed review of convective heat transfer and aerodynamics in axial flow turbines is now available by Dunn (Han et al., 2012).

**Coolant passage heat transfer with rib turbulators**

In advanced gas turbine blades, rib turbulators are often cast on two opposite walls of internal coolant passages to augment heat transfer. The internal coolant passages are mostly modelled as short, square or rectangular channels with various aspect ratios.

The heat transfer augmentation in rectangular coolant passages with rib turbulators primarily depends upon the rib turbulators’ geometry, such as rib size, shape, distribution, flow-attack-angle, and the flow Reynolds number. Rib turbulators disturb only the near-wall flow for heat transfer enhancement. Therefore, the pressure drop penalty caused by rib turbulators is affordable for the blade internal cooling designs. There have been many basic studies by Han et al. (Taslim and Bethka, 2008) and Han to understand the heat transfer augmentation versus the pressure drop penalty by the flow separation caused by rib-turbulators. Figure 12.3.5 (a) shows symmetric (parallel) and staggered rib arrangements in opposite walls of a cooling channel. Figure 12.3.5 (b) shows schematics of flow separations from repeated ribs reattach on the floors between ribs thus increasing the heat transfer coefficients of the ribs and the floors.

The heat transfer coefficients can be further enhanced by casting the ribs with an angle to the coolant flow, which causes a rib-induced secondary flow moving in the rib angle direction. The Reynolds numbers based on coolant channel hydraulic diameter vary from 10,000 to 80,000. However, the Reynolds numbers can be up to 500,000 for the coolant passages in large power generation turbine blades. In general, the repeated ribs that used for coolant passages with a channel aspect ratio varying from 1/4 (near blade leading edge) to 4 (near blade trailing edge), are nearly square in cross section with a typical relative rib height of 5–10% of the coolant channel hydraulic diameter, a rib spacing-to-height ratio varying from 5–15, and a rib flow-attack-angle around 30–60°. In general, smaller rib height is more efficient for higher Reynolds number flows, and the heat transfer enhancement decreases but pressure drop penalty increases with the Reynolds number. For example, the heat transfer can be enhanced about 3 times with 5 times pressure drop penalty in a square channel with typical rib geometry (6% rib height-to-channel hydraulic diameter ratio, 10 rib spacing-to-height ratio, and 45° rib flow-attack-angle) at a Reynolds number around 30,000.
Several studies by Han et al. (Taslim and Bethka, 2008) show that the V-shaped and Delta-shaped ribs provide better heat transfer performance than the typical angled rib geometry for a given pressure drop penalty. Figure (12.3.6) shows conceptual view of secondary flow vortices induced by angled ribs and V-shaped ribs. However, larger rib
height-to-channel hydraulic diameter ratio can be used to generate a little more heat transfer enhancement if the pressure drop penalty is not a main concern in some highly demanding cooling designs. Also, the closer (or wider) rib spacing has reduced heat transfer enhancement and pressure drop penalty.

![Diagram of rib arrangements](image)

Figure 12.3.5. (a) Symmetric and staggered rib arrangements in opposite walls of cooling channel. (b) Schematic of flow separation from ribs and secondary flow between angled ribs in a rib-turbulated cooling channel.

Therefore, the closer (or wider) rib spacing can be used to get enough heat transfer enhancement if the pressure drop penalty is a major concern in certain cooling designs (Han, 2004). In these cases, the closer rib spacing can be used due to the increased rib-side area for convection (fin effect), in addition to the heat transfer enhancement. For example, smaller gas turbine blades have larger blockage ribs with 10–20% rib height-to-hydraulic diameter ratio at closer spacing with 3–5 rib spacing-to-height ratio reported by Taslim et al. (Han, 2004).
12.3.1. Correlations of film cooling performance

The primary measure of film cooling performance is the film effectiveness, $\eta$, since this has a dominating effect on the net heat flux reduction. Furthermore, industrial designers typically will focus on the laterally averaged film effectiveness $\bar{\eta}$, which is the average $\eta$ over a line normal to the flow and extending a distance equal to the pitch between holes. Besides the simplification in processing film effectiveness results by using only laterally averaged data, there is a physical rationale for using only the laterally averaged film effectiveness.

For the baseline case described above, the coolant holes were angled such that the exiting coolant jets are parallel with the mainstream direction. When the coolant hole is angled to the mainstream direction, this is referred to as “compound angle” injection. Compound angles can be as much as 90°, i.e. normal to the mainstream direction. Coolant injected at a compound angle is quickly turned to the mainstream direction, but will generally have a broader distribution of coolant. Furthermore, the coolant presents a broader profile to the mainstream so that the mainstream has a larger impact on the jet more effectively turning the jet towards the wall. This inhibits jet separation, and results in
better film effectiveness for the compound angle holes at higher blowing ratios. Film effectiveness performance for 90° compound angle holes compared to 0° (streamwise oriented holes), shown in Figure (12.3.7), illustrates this point. These data are for cylindrical holes spaced 6.5d apart on a smooth flat test surface with low mainstream turbulence levels. Maximum film effectiveness for the 90° compound angle holes was similar to that for the 0° holes and occurred at a similar momentum flux ratio. However the 90° compound angle holes sustained high film effectiveness for very high blowing ratios. For momentum flux ratios greater than \( I = 1.0 \), the film effectiveness for the 90° compound angle holes was a factor of 2 to 3 higher than that for the streamwise-oriented holes. Although the film effectiveness for compound angle holes is significantly better than for streamwise-oriented holes at high momentum flux ratios, the net heat flux reduction for compound angle holes is similar to the streamwise-oriented holes 17. This is illustrated in Figure (12.3.8) for 90° compound angle holes. At the higher momentum flux ratio of \( I = 1.1 \) the average \( \Delta q_r \) over the 90d distance downstream of the coolant holes was about the same for 90° and 0° compound angle holes.

![Figure 12.3.7](image)

Figure 12.3.7. Comparison of streamwise and laterally directed holes in terms of laterally averaged effectiveness as a function of momentum flux ratio for a smooth surface and low free-stream turbulence

The similarity of the net heat flux reduction even though the film effectiveness is much greater for 90° compound angle holes is due to a greater increase in heat transfer coefficient for these holes compared to streamwise-oriented holes. Even though the
average increase in heat transfer coefficient by the compound angle holes was only 10%, this was sufficient to offset the improved film effectiveness (Goldstein, 1971).

![Graph showing comparison of streamwise and laterally directed holes in terms of net heat flux reduction for a smooth surface and high free-stream turbulence.]

**Film Cooling with Shaped Holes**

Improved film effectiveness can be achieved if the exit of the hole is expanded so that coolant is slowed through a diffuser. Examples of shapes investigated in the open literature are shown in Figure (12.3.9). There are two advantages for such a “shaped hole”: the coolant exit velocity is reduced and a broader jet cross-section is presented to the mainstream flow.

Both these characteristics will reduce the tendency for the coolant jet to separate. This results in good film effectiveness levels for shaped holes at very high blowing ratios as shown in Figure (12.3.10). These data were obtained with a row of coolant holes angled 30° with the surface and spaced 4$d$ apart. The spatially averaged film effectiveness, $\eta$, was based on a average from $x/d = 2$ to 22. The blowing ratio for this figure is based on the average velocity of the coolant at the inlet to the coolant hole, so the mass flow of coolant for the cylindrical and shaped holes are the same for the same $M$. Film effectiveness for cylindrical holes begins to decrease for $M > 0.7$ which corresponds to a momentum flux ratio of $I >0.3$ given that the density ratio for these tests was $D_R = 1.7$ (Bogard, 2002).
Figure 12.3.9. Schematics of different cooling hole shapes (reproduced with permission from Journal of Turbomachinery)

Figure 12.3.10. Comparison of spatially averaged cooling effectiveness for cylindrical holes and shaped holes (reproduced with permission from Journal of Turbomachinery)
This decrease is due to separation of the coolant jets. In contrast the film effectiveness for the shaped holes continues to increase for blowing ratios up to $M = 2.5$ ($I = 3.7$) showing that the diffusing hole shape is very effective in keeping the coolant jets attached (Colban and Thol, 2006).

Surface curvature and surface roughness are significant factors affecting film cooling performance. Clearly for turbine airfoils strong convex curvature exists around the leading edge and along the suction side of the airfoil. Sometimes strong concave curvature is encountered on the pressure side of the airfoils. Surface roughness varies with the length of operation of the engine; new airfoils are relatively smooth, but after some period of operation the surfaces can become quite rough due to erosion, spalation of thermal barrier coatings, and deposition of contaminants. The following is a brief review of these surface effects.

**Surface curvature**

Several studies have shown that surface curvature can significantly change film effectiveness; convex curvature increasing $\eta$ and concave curvature decreasing $\eta$ at typical operational blowing ratios. The effects of varying strengths of curvature are demonstrated in Figure (12.3.11) in which the laterally averaged film effectiveness $\eta$, at $x/d = 40$ are presented for a range of curvatures, $46 < 2r/d < 126$, with zero pressure gradient ($r$ is the radius of curvature for the surface). These studies indicated that an increased convex curvature (decreasing $2r/d$) greatly enhances film effectiveness, while concave curvature decreases film effectiveness except at high momentum flux ratios. These effects of surface curvature can be explained by the wall normal pressure gradients that necessarily exist with wall curvature. When the momentum of the jet tangential to the wall is less than the mainstream momentum the normal pressure gradients drive the coolant jets towards or away from the wall for convex and concave curvature, respectively. For convex curvature, the inward pressure broadens the coolant distribution by pressing the jet to the wall, and keeps the jet attached for higher momentum flux ratios. For concave curvature the opposite occurs, i.e. the coolant jets are pushed away from the wall.

**Surface Roughness**

Significant increases in surface roughness during typical operating conditions have been reported by several studies, with maximum roughness levels as high as $R_{ek} = 500$ where $R_{ek}$ is the equivalent sandgrain roughness Reynolds number. Given that “fully rough” conditions exist when $R_{ek} > 70$, this roughness level is extremely large. Also, maximum roughness heights were observed to greater than 250 $\mu$m, which is $0.5d$ for
typical coolant hole diameters. Surface roughness degrades film cooling performance by increasing the heat transfer coefficient and potentially reducing film effectiveness. Heat transfer coefficients can be increased by as much as 50% to 100%. Studies of the effects of surface roughness on film effectiveness using flat surface facilities showed small reductions (<10%) of average film effectiveness for lower blowing ratios, and small increases for high blowing ratios. However, a study of roughness effects on film effectiveness on the suction side of a vane showed surface roughness decreased film effectiveness by as much as 25% at the optimum blowing ratio, but increased film effectiveness as much as 50% at higher blowing ratios. The decrease in film effectiveness at the optimum blowing ratio was primarily due to the roughness upstream of the coolant holes. The upstream roughness doubled the boundary layer thickness and significantly increased turbulence levels which resulted in more separation of the coolant jets and increased dispersion of the coolant.

Figure 12.3.11. Effect of convex and concave curvature on film effectiveness (reproduced with permission from Journal of Turbomachinery)

There are a number of mainstream factors that can affect film cooling performance including approach boundary layers, turbulence levels, Mach number, unsteadiness, and rotation (Bogard, 2002). Because of the very high levels of mainstream turbulence exiting
the combustor and entering the turbine section, turbulence levels have the largest effect on film cooling performance. Mainstream turbulence levels exiting the combustor can be higher than $Tu = 20\%$ and have been found to be nominally isotropic in simulated combustor studies. Furthermore the integral length scale of the turbulence is large relative to the coolant hole diameters, i.e. $\Lambda f/d > 10$ (based on $\Lambda f$ values given in Radomsky and Thole). Primarily due to the acceleration of the mainstream as it passes around the first vane, the local turbulence levels reduce to less than 5% on the suction side of the vane, and to about 10% for much of the pressure side. These are still relatively high turbulence levels, and it is important to recognize the effects on film cooling performance. High mainstream turbulence levels degrade film cooling performance by increasing heat transfer coefficients and generally decreasing film effectiveness. Simulations of the large scale turbulence with levels of $Tu = 10\%$ to 17% showed an increase in heat transfer coefficient of 15% to 30%, respectively. The effects of high mainstream turbulence levels on film effectiveness are shown by the laterally averaged film effectiveness levels for $Tu = 0.3\%, 10\%, \text{and } 20\%$ shown in Figure (12.3.12). Results in Figure (12.3.12) were obtained using a flat surface test facility with a row of cylindrical holes spaced $6.5d$ apart, with an injection angle of $30^\circ$ and aligned with the mainstream direction. Smooth and rough surfaces were tested. The coolant density ratio was $DR = 2.0$. For a smooth surface with low turbulence levels the optimum momentum flux ratio was $I = 0.3$. At this momentum flux ratio, a turbulence level of $Tu = 17\%$ caused a factor of two decrease in film effectiveness near the hole, and almost a complete loss of cooling for $x/d > 25$. The optimum momentum flux ratio for high mainstream turbulence conditions was about $I = 1.1$, substantially higher than would have been expected from low mainstream turbulence tests. At this higher momentum flux ratio the film effectiveness for the high mainstream turbulence case was higher than for the low mainstream turbulence case.

This difference was attributed to the higher mainstream turbulence mitigating the effect of coolant jet separation by returning some of the coolant towards the surface with the increased coolant dispersion caused by the higher turbulence levels. These results show the importance of accounting for realistic mainstream turbulence levels when predicting film cooling performance (Bogard, 2002).
12.4. A conjugate heat transfer method for turbine blade cooling

In general, the study of heat transfer gives useful information to provide efficient solutions to many common engineering problems, as heating in circuits, heat exchanger or gas turbines. For instance, modern turbines reach temperatures which exceed the melting point of the blade material.

Conjugate heat transfer (CHT) is the process regarding the interaction between the heat conduction inside the solid body and the heat transfer in the surrounding fluid. In real problems, the near wall flow is highly influenced by the solid thermal status.
In the external blade cooling, cold air is injected through the film cooling holes on the external blade surface in order to create a thin film cooling layer (Figure 12.4.1). In the internal method, the heat is removed by a variation of convection and impingement cooling configurations, where high velocity air flows and hits the inner surfaces of the turbine vanes and blades.

Conjugate heat transfer (CHT) boundary conditions determine by the interface fluid temperature is known from the previous step, the temperature can be calculated at the fluid interface cells. After grid generation, normal vectors and distances along the normal have been calculated for each interface cell.

As a result of the conjugate calculation approach, the surface temperatures and temperature fields in the nozzle solid body can be determined. Figure (12.4.2) shows the external surface temperature distribution on the nozzle vanes pressure side and shrouds, and Figure 3 on the suction side and shrouds, respectively. The damage to the nozzle was evaluated based on the analysis by Mazur et al. (2006).
The maximum temperature (934 °C) on the pressure side is localized at the central part of the vane airfoil (50% height) on the leading edge and decreases gradually in a direction perpendicular to the vane axis, reaching a local minimum at the film cooling ducts (640.5 °C). Next, the metal temperature is increasing at trailing edge and reaching a local maximum (853.9 °C).

Similarly, the maximum temperature (934 °C) on the suction side is localized at the central part of the vane airfoil (50% height) on the leading edge. Then the temperature decreases gradually in a direction perpendicular to the nozzle axis, and drops abruptly at the two rows of film cooling ducts, near the vane leading edge until it reaches a local minimum (640.5 °C).
Figure 12.4.3. External surface temperature distribution on the nozzle vane suction side and shrouds.

Afterwards, the temperature increases at the trailing edge, reaching a local maximum of 853.9 °C. Figure (12.4.4) gives the distribution of internal vane temperatures in the cutting plane at 50% height (section of maximum temperature). As it can be appreciated, due to relatively thin vane walls, the temperature gradients in the solid hollow body of the vane are not too high.
The diagram of the temperature distribution on the external surface of the nozzle vane transversal section in the cutting plane at 50% height is shown in Figure (12.4.4). As it can be seen in Figs. (12.4.2), (12.4.3) and (12.4.4), the temperature gradients in the film cooling ducts regions are very high and these zones must be taken into account in a
succeeding thermal stress analysis. Fig. (12.4.5) shows the same temperature distribution on the external surface of the nozzle vane transversal section in the cutting plane at 50% height varying cooling air flow rate and temperature. For the nominal cooling flow rate of 4% (related to the principal gas flow) the nozzle metal maximum temperature is 934 °C (see pink curve). Increasing air flow rate to 8%, the maximum nozzle metal temperature is reduced to 860 °C (navy-blue curve). Increasing nominal air flow temperature by 100 °C and maintaining nominal air flow rate, the maximum metal temperature is increased to 950 °C (red curve).

Figure 12.4.5. External surface temperature distribution on the nozzle vane transversal section in the cutting plane at 50% height for varying cooling air flow rate and temperature

Finally, increasing air flow temperature by 200 °C for increased air flow rate of 8%, it gives metal maximum temperature of 975 °C (green curve) (Mazur et al., 2006).

As it can be appreciated, the cooling air flow rate and temperature influence significantly the heat transfer in the nozzle solid body and results in metal maximum temperature and temperature profile. As a result of this investigation, the predicted nozzle temperature distribution made it possible to numerically analyse the thermal stresses and creep loads on the nozzle in steady state operation of the gas turbine. Steady-state analyses of conjugate heat transfer of a first stage nozzle were conducted in order to predict the temperature distribution during continuous load operation. In order to simulate the actual nozzle, the ejection of the internal cooling air into the gas path, the distribution of the bulk temperature of the cooling air in the internal passage, and the inlet distribution of gas
temperature were integrated into the computational model. As a result, the computations were able to simulate the heat transfer in the nozzle during steady-state operation. The conducted investigation shows that the cooling air flow rate and temperature influence significantly the heat transfer in the nozzle solid body and, as a result, the metal maximum temperature and temperature profile. Also, the assessment of service induced degradation of cobalt base alloy FSX-414 of the nozzle, after 24,000 h of operation at high temperature was carried out. Temperature distribution on the nozzle vane (Figs. 12.4.2, 12.4.3, 12.4.4 and 12.4.5), it is confirmed that a direct relation between the degree of alloy deterioration and metal temperature exists. It is noteworthy that the largest grain size and volume fraction of carbides corresponds to the highest metal temperature zones. This is one of the most of important facts found in this investigation. The present predictions made it possible to estimate stress and creep strain loaded on the nozzle and to assess the nozzle remaining life based on the evaluation of the damage.
References

References in English:


Kercher D. and Tabakoff W. Heat transfer by a square array of round air jets impinging perpendicular to a flat surface including the effect of spent air. J. of Enggr. for Power, vol.92, 1970, pp. 73-82.


References in Russian:
Григорьев Б.А., Цветков Ф.Ф. Тепломассообмен. Москва: МЭИ. 2011, 562 с.
Чугаев Р.Р. Гидравлика, 4-е изд. ленинград: энергоиздат, 1982, 672 с.
Кейс В.М. Конвективный тепло- и массообмен. Москва: Энергия. 1972, 448 с.
Коловандин Б.А. Моделирование теплопереноса при неоднородной турбулентности. Минск: Наука и техника, 1980, 184 с.
Колмогоров А.Н. Локальная структура турбулентности в несжимаемой вязкой жидкости при очень больших числах Рейнольдса. ДАН СССР. 1941, Т. 30, №. 4, с. 299–303.
Куини Т. Температура. Москва: Мир. 1985, 448 с.
Лыков А.В. Теоретические основы строительной теплофизики. Минск: Академия наук БССР. 1961, 519 с.
Патанкар С. Численные методы решения задач теплообмена и динамики жидкости. Москва: Энергоатомиздат. 1984, 152 с.
Пономарев С.В., Мищенко С.В., Дивин А.Г. Теоретические и практические аспекты теплофизических измерений. Книга 1. Тамбов: Тамбовский государственный технический университет. 2006, 204 с.
Томпкинс У., Уэбстер Дж. Сопряжение датчиков и устройств ввода данных с компьютерами IBMPC. Москва: Мир. 1992, 592 с.
Трофимов А.И. Приборы и системы контроля ядерных энергетических установок. Учебное пособие. Москва: Энергоатомиздат 1999, 494 с.
Уайт Г.К. Экспериментальная техника в физике низких температур. Москва: Наука. 1961, 368 с.

Internet Resources:
FLUENT User Forum. CFD Online - Discussion Forums http://www.cfd-online.com/Forum/fluent.cgi
ICEM. Dassault Systemes homepage http://www.icem.com
PHOENICS Overview. CHAM http://www.cham.co.uk/phoenics/d_polis/d_info/phover.htm
Star-CD.CADFEM http://www.cadm.com/program/star_cd/star_cd.htm
TNF Workshop web site http://www.ca.sandia.gov/TNF