

KompasFlow

**KOMPAS-3D integrated tool for hydrodynamics and
thermodynamics express analysis**

**Document created at 28.01.2026.
Relevant for KompasFlow version 24.1.0**

Table of contents

1 Purposes and capabilities of KompasFlow	5
2 Before use	6
3 Basic elements of the KompasFlow interface	7
4 Quick start training case: mixing hot and cold water	9
4.1 Creating a KompasFlow project	9
4.2 General settings	12
4.3 Computational domain geometry	13
4.4 Substance and its parameters	14
4.5 Physical processes	15
4.6 Boundary conditions	16
4.7 Initial conditions	24
4.8 Computational grid	25
4.8.1 Initial computational grid	25
4.8.2 Adaptation	26
4.9 Parameters of calculation controls	28
4.10 Viewing results of the computation	29
4.10.1 Creating layers	29
4.10.2 Creating results	33
4.10.3 Starting the project's computation	36
4.10.4 Viewing visualization layers during the computation	37
4.10.5 Data viewing in the Monitoring window	38
5 Interface reference guide	40
5.1 KompasFlow toolbar	41
5.2 Wizard	44

5.3 Templates	44
5.4 Items in the project window	46
5.4.1 Region	47
5.4.2 General settings	47
5.4.2.1 Simulating an external flow	48
5.4.3 Computational domain geometry	49
5.4.4 Substance	51
5.4.5 Physical processes	53
5.4.6 Boundary conditions	54
5.4.6.1 Symmetry	57
5.4.6.2 Wall	58
5.4.6.3 Inlet/Outlet	60
5.4.6.4 Free outlet	65
5.4.6.5 Non-reflecting	66
5.4.6.6 External BC	67
5.4.6.7 Source of moisture	67
5.4.6.8 Hood	69
5.4.7 Initial conditions	70
5.4.8 Computational grid and adaptation	71
5.4.9 Computation parameters and limiters	74
5.4.10 Visualization layers	76
5.4.10.1 General properties of layers	77
5.4.10.2 Color contours	81
5.4.10.3 Vectors	83
5.4.10.4 Streamlines	83
5.4.10.5 Computation grid section	85

5.4.10.6 Isosurface	86
5.4.11 Results (folder).....	86
5.4.11.1 Results (items).....	87
5.5 Monitoring window	88
5.6 Changing a project connected to the solver	94
5.7 Optimization	95
5.8 Creating reports	98
6 Problem solving	100
6.1 Possible problems.....	100
6.2 Error messages and warnings	100
6.3 Technical support.....	101

1 Purposes and capabilities of KompasFlow

Purposes and capabilities of KompasFlow

The KompasFlow application is a KOMPAS-3D integrated tool for rapid testing of the aerohydrodynamics of a design device.

KompasFlow has a simple interface for rapid testing of a device in early design stages and makes it possible to make a primary assessment of the impact of modifications of device geometry on its effectiveness.

KompasFlow can simulate:

- Single-component gas flows (supersonic and subsonic flow, compressible and incompressible media).
Examples of tasks:
 - Calculation of the aerodynamic drag of a vehicle, wing lift
 - Flow in ventilating ducts and through ventilating grilles, calculation of losses
 - Air circulation in rooms, cabins
 - Calculation of wind load on structures and constructions
- Flows of liquids. Examples of tasks:
 - Calculation of hydrodynamic losses in pipes and valves
 - Calculation of hydraulic losses in liquid heat exchangers
- Thermal conductivity and natural convection with account of radiant heat exchange. Examples of tasks:
 - Simulation of heat removal in heat exchangers
 - Simulation of ventilation, cooling and heating of rooms and cabins
 - Analysis of cooling efficiency in electrical engineering.

KompasFlow supports parallel calculations within one multicore processor.

A simple interface imposes constraints on the range of tasks:

- flow and heat transfer are simulated in only one enclosed volume
- the calculated volume can be occupied by only one substance
- the first versions of the program do not provide for simulation of moving objects.

Each version of KompasFlow will be developed to solve an increasingly wide range of tasks.

Integration with the FlowVision general purpose package of computational aerohydrodynamics

KompasFlow hydrodynamic rapid test module uses the FlowVision universal hydrodynamic package. If you need to solve complex tasks with multicomponent substances, chemical reactions and complex movements, you should use FlowVision.



Projects prepared in KompasFlow can be opened in FlowVision, where you can continue the calculation with a more complex and accurate task formulation.

2 Before use

Before you start use KompasFlow, you should complete the following steps.

Setting connection to Solver

Solver (fvsolver.exe or fvsolver64.exe) creates a TCP server for a socket connection with KompasFlow. When you start Solver, KompasFlow tells it, on which port a TCP server is to be created and is to be listened for the connection.

Ports 39900-39999 are used. Starting from port 39900, there is a search for free ports, which is necessary for several solvers to be launched for different tasks.

The connection between Solver and KompasFlow is internal, and there are neither calls to the outside nor calls to the local and global networks. Even if a Windows' antivirus software and/or a firewall reports about an attempt of access to closed ports from the specified range, this will not prevent data exchange between Solver and KompasFlow. You will not see such message if you open access to ports in the range 39900-39999.


Plugging KompasFlow in KOMPAS-3D

The KompasFlow application is a part of the standard KOMPAS-3D distribution kit (starting from version 18), but it is not plugged in by default. To plug it in, open the KOMPAS-3D's Configurator's using the main menu command **Applications > Configurator**.

If you wish KompasFlow be plugged in automatically each time you start KOMPAS-3D, select the check box Auto Connect at System Startup. Then click Connect.

If, for some reason, the KompasFlow application has been excluded from the configuration, you can add it to the configuration again using the command Configuration > Add applications from the Configurator's menu or command Applications > Add applications from the KOMPAS-3D's main menu. In the form, which opens, you should select the FvKompasPlugin.rtw file (for version 23 of KOMPAS-3D, this file locates by default in folder C:\Program Files\ASCON\KOMPAS-3D v23\Libs\KompasFlow). After successful plugging KompasFlow in, you will see a message about this.

First running KompasFlow for calculation

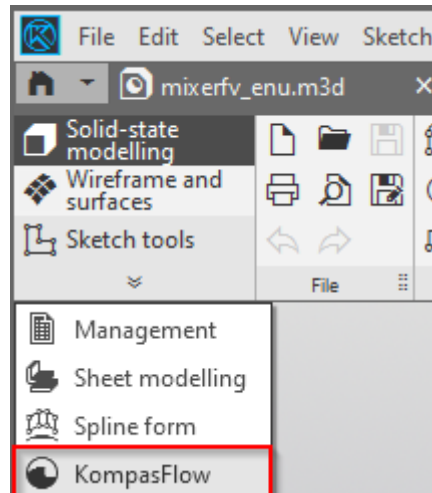
 After installing KompasFlow, its first running for calculation can cause a delay of more than a minute.

3 Basic elements of the KompasFlow interface

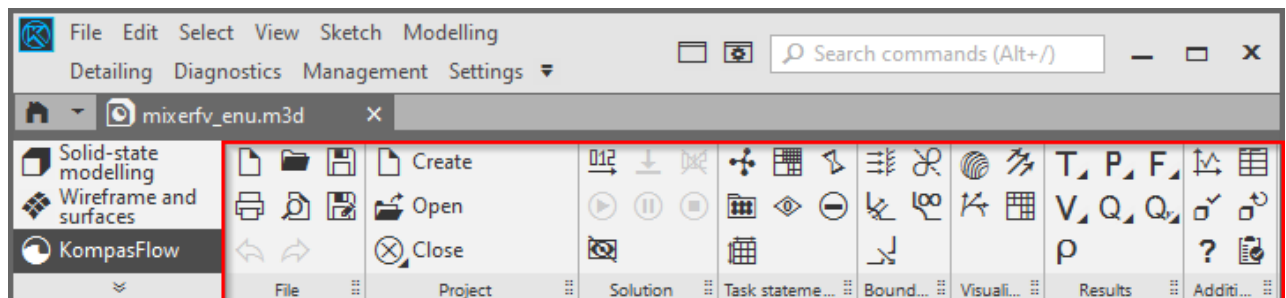
KompasFlow is the library of the three-dimensional simulating system KOMPAS-3D, and uses its user interface (graphical area, control panel, property toolbar, screen tips, informational messages). See the [section of the KompasFlow interface handbook](#).

KompasFlow toolbar

After connecting to KompasFlow, open the KompasFlow toolbar by selecting it from the list of available toolbars:

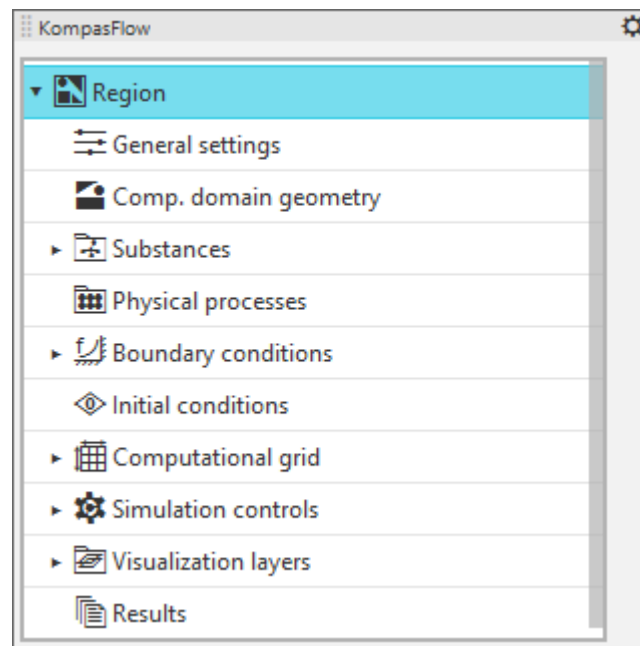


KompasFlow [toolbar](#) opens at the top of the KOMPAS-3D window:



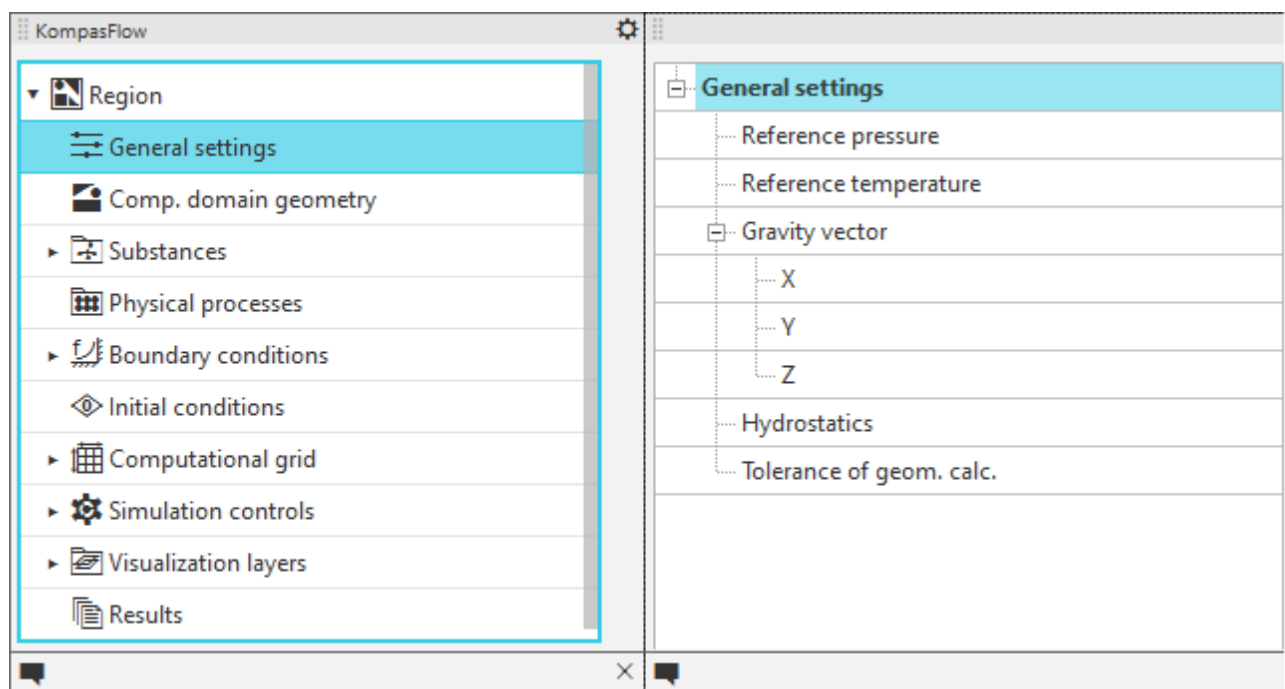
KompasFlow project tree

KompasFlow elements are presented in the KOMPAS-3D control panels, in the KompasFlow tab. We refer them as "project tree", not confusing them the geometric model tree of KOMPAS-3D.



Property bar

Parameters of the selected KompasFlow element are displayed and can be set in the property bar:

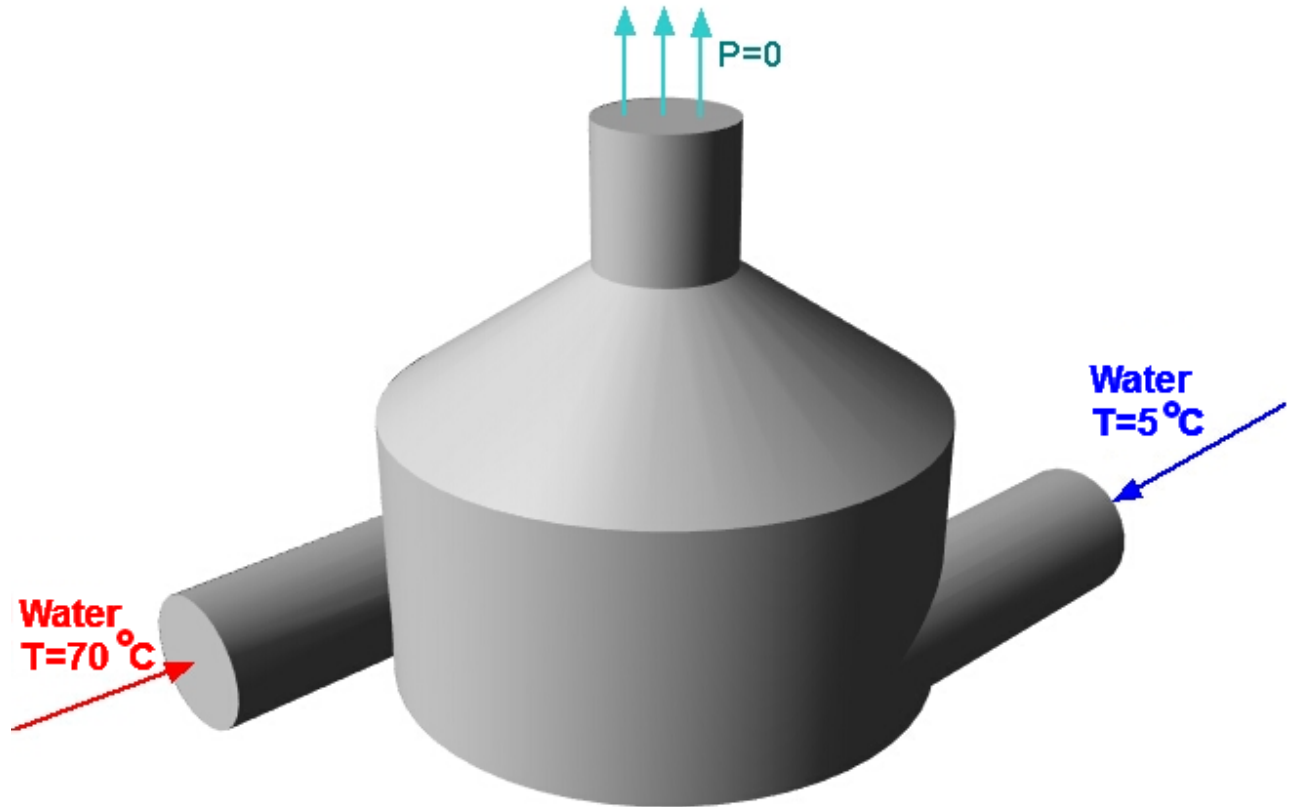


Parameter values in the properties bar are either entered in text fields or selected from the drop-down list.

4 Quick start training case: mixing hot and cold water

As a step-by-step case for your basic training with KompasFlow, we consider a flow of liquid in a mixer.

We consider flow of water in a mixer that consists of a chamber with two inlet tubes near its base and one outlet tube at its top:



The diameter of the chamber is 0.04 m, the height of the cylindrical part of the chamber is 0.02 m, the height of the cover is 0.01 m, the diameter of the tubes is 0.01 m.

Cold water is fed through one of the tubes ($T = 5\text{ }^{\circ}\text{C}$). Hot water ($T = 70\text{ }^{\circ}\text{C}$) is fed through the other. The flow mass rate in the inlet tubes is the same and equal to 0.1 kg s^{-1} .

The purpose of the calculation is to visualize the flow and equalization the water temperature in the mixer.

4.1 Creating a KompasFlow project

Geometry of the computational domain, on which this training project is based, has already been created and it is stored in the mixerfv_enu.m3d file.

The mixerfv_enu.m3d file, which is included in the KompasFlow package, is used as a geometric model for the training case (by default, for version 23 of KOMPAS-3D this file locates in folder C:\Program Files\ASCON\KOMPAS-3D v23\Libs\KompasFlow\Samples).

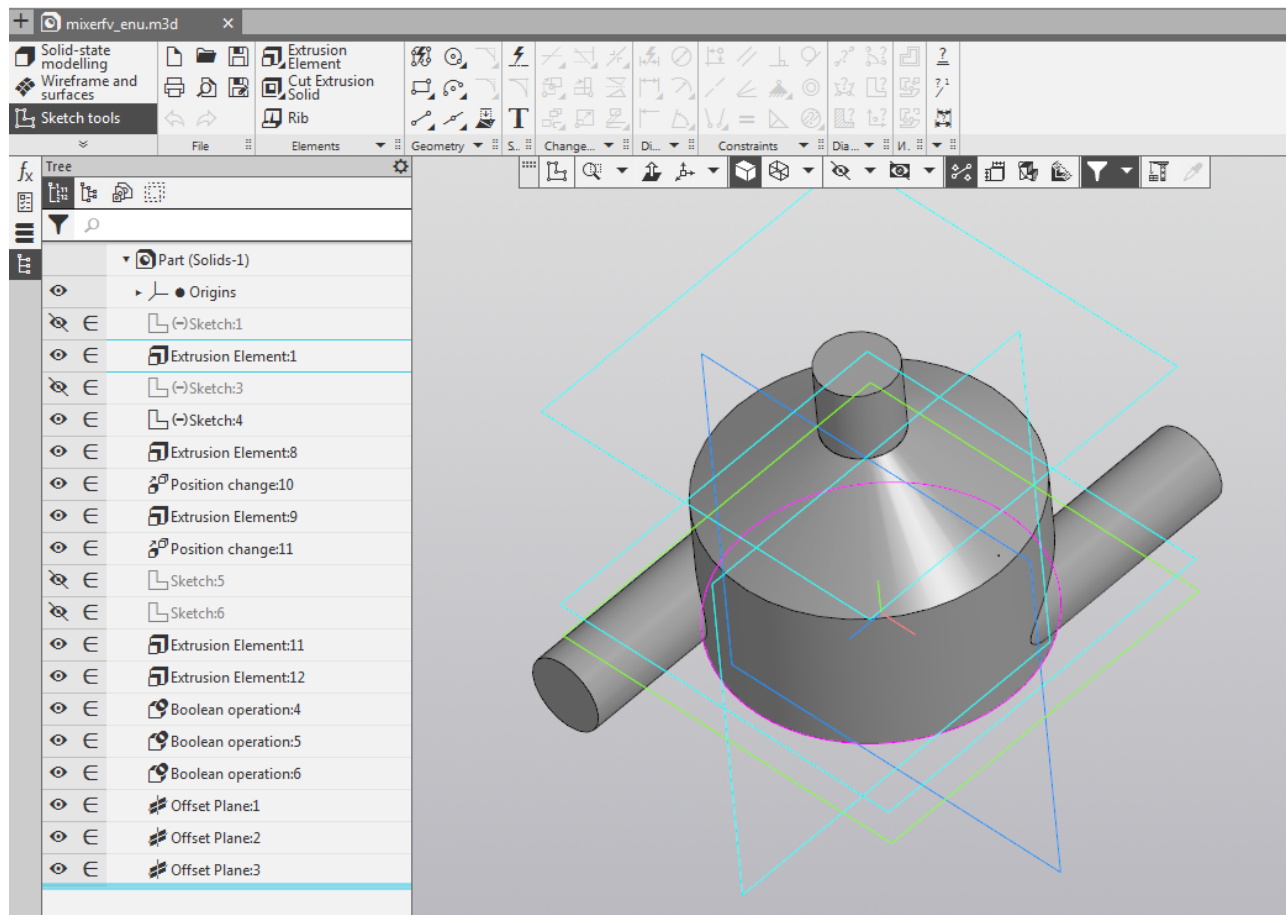
Copy it to any folder that can be written without administrator rights (in order not to damage mixerfv_enu.m3d source file accidentally and since Windows systems folders settings such as ProgramFiles may not give the write rights needed to work with the geometrical model).

To load it into the KompasFlow application, follow these steps:


1. Open the geometrical model of the computational domain

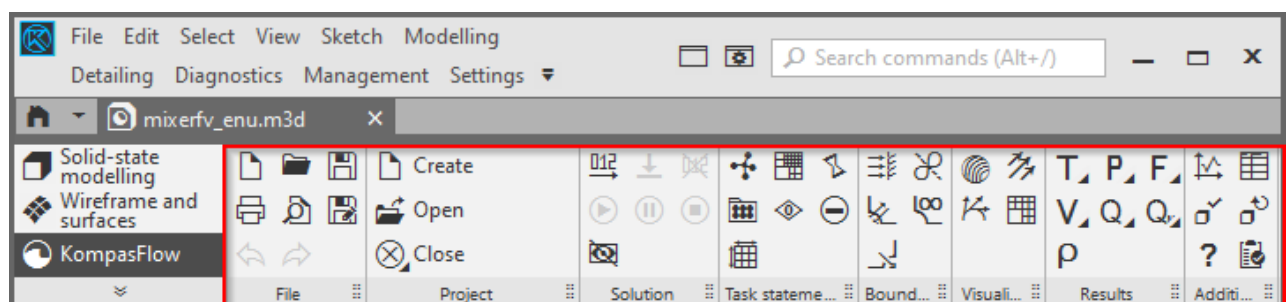
Load the geometrical model of the mixer from the KOMPAS-3D main menu, apply the command  File > Open and select the file in the new window.

The geometrical model of the mixer will be displayed in the graphical area of the KOMPAS-3D window:



2. Open the KompasFlow toolbar

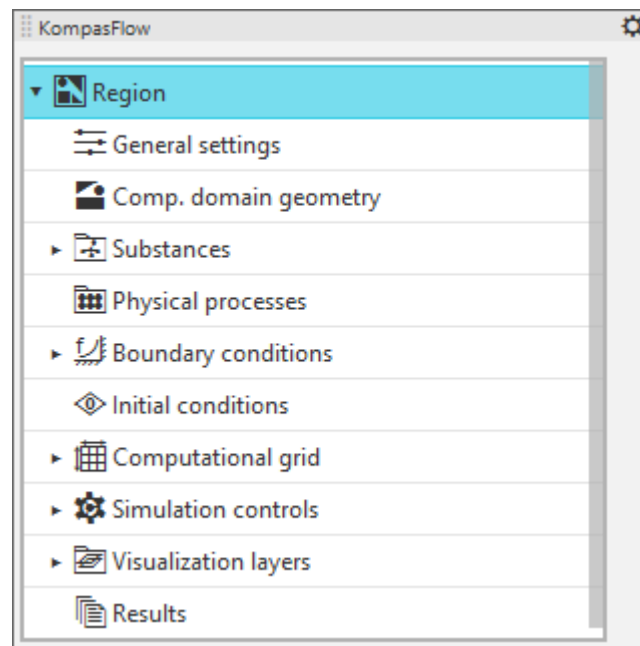
In the list of toolbars, open the [KompasFlow toolbar](#) (if necessary, expand the list by clicking the symbol ):



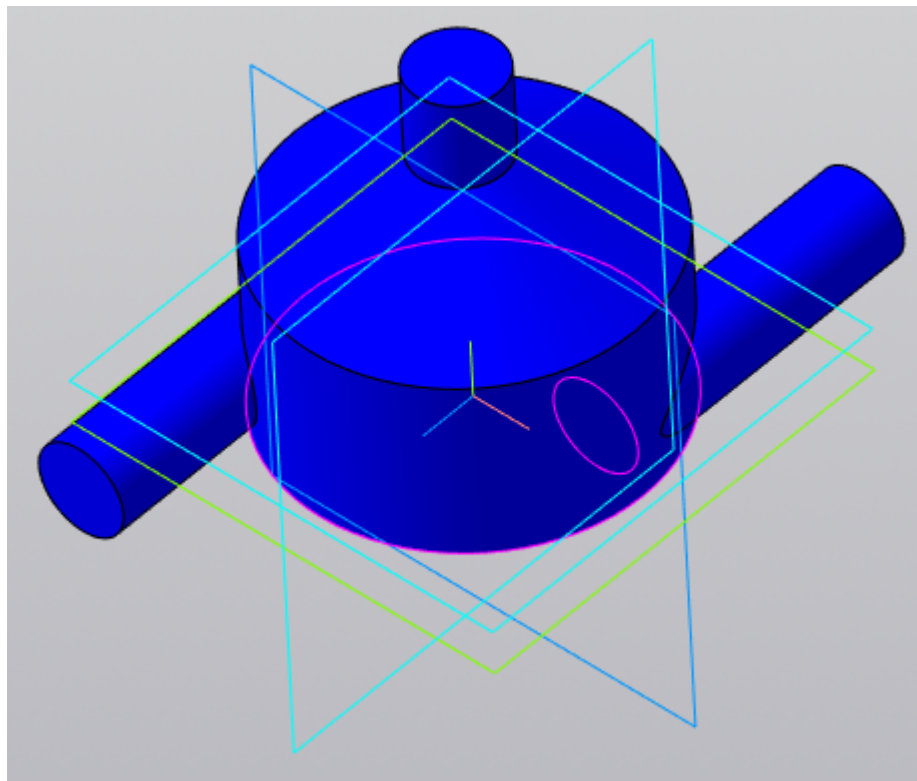
3. Create the KompasFlow calculation project.

In the [KompasFlow toolbar](#), click  **Create**.

In the KOMPAS-3D control panels, a new KompasFlow tab will appear (KompasFlow's project tree):

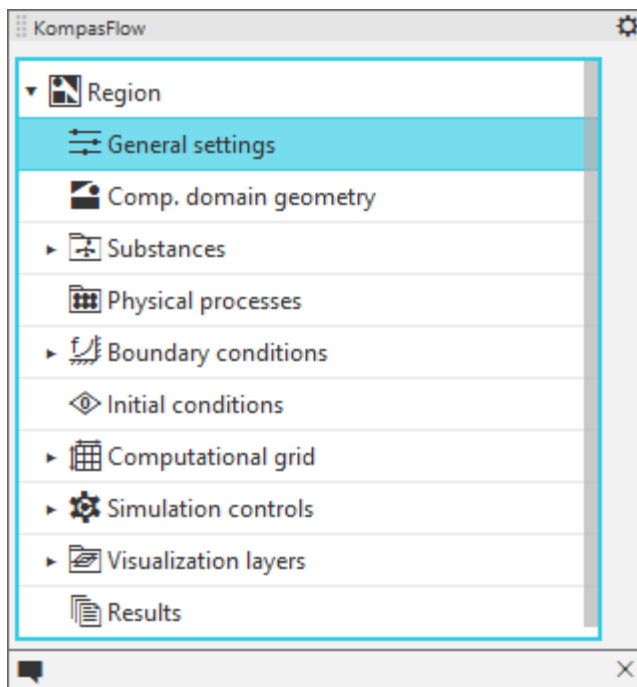


Now a new KompasFlow computation project is created.



4.2 General settings

In **General settings** of a project, the gravity vector, the reference pressure and temperature, and geometric accuracy are specified.

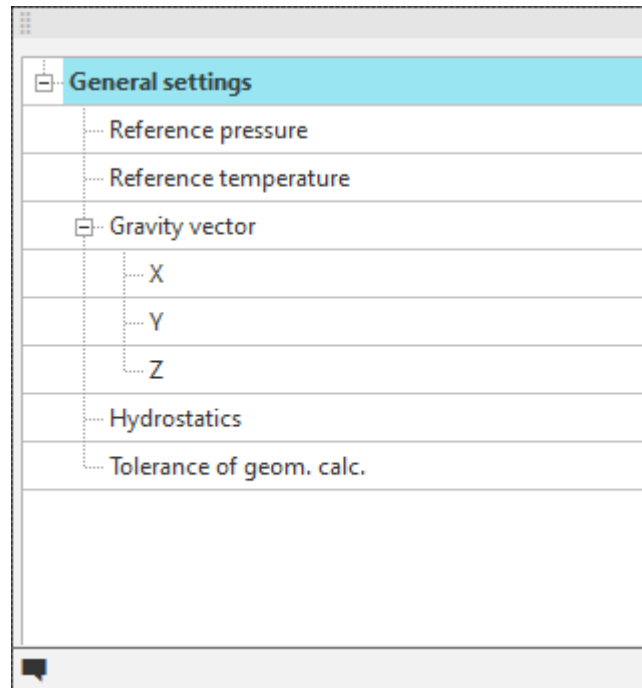


For this example, gravity has negligible influence on the solution, so the default zero values of the Gravity Vector components should be retained.

Global parameters are set in the properties bar. Details regarding global parameters are provided in the dedicated chapter.

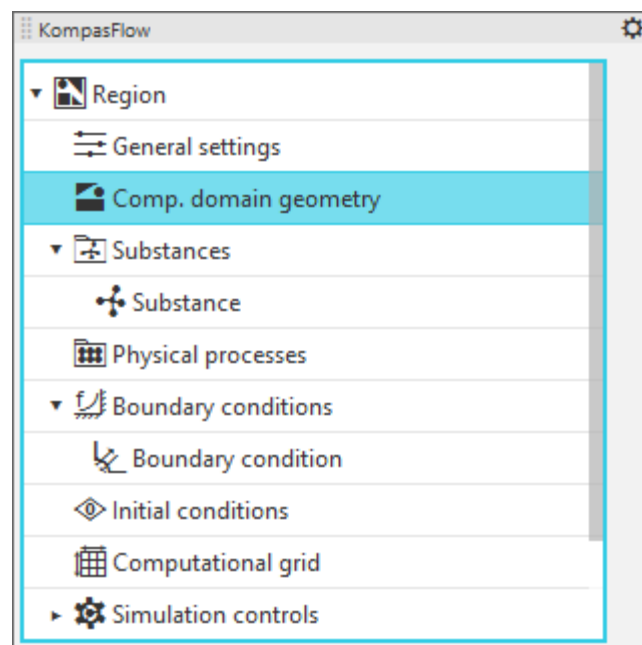
The following default settings are recommended for this case:

Reference pressure	101000
Reference temperature	273
Gravity vector	
X	0
Y	0
Z	0
Hydrostatics	No
Tolerance of geom. calc.	1e-08
External task	No
BC external region	Nonreflect



4.3 Computational domain geometry

The KompasFlow project tree includes, among others, the folder [Comp. domain geometry](#), with subfolder Bodies:

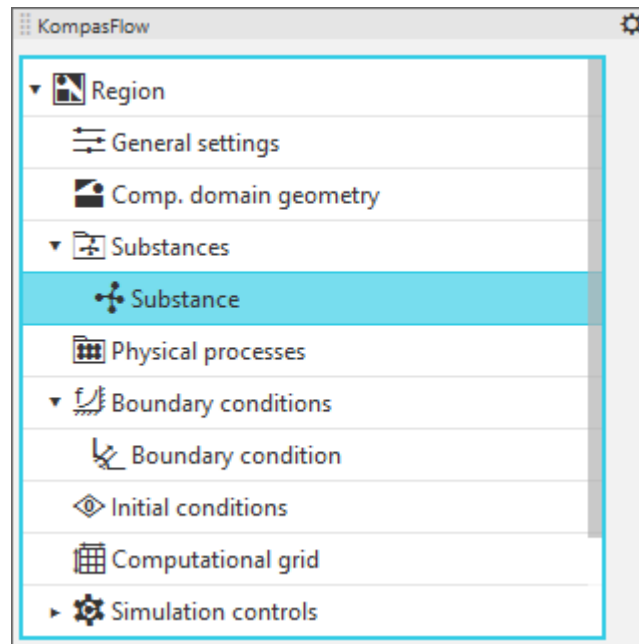


If the body consists of several parts, all parts will be displayed in this folder. Here you can determine which body will be the main calculated volume, and which will be the insert that changes the calculated volume.

Bodies folder is not used in this tutorial.

4.4 Substance and its parameters

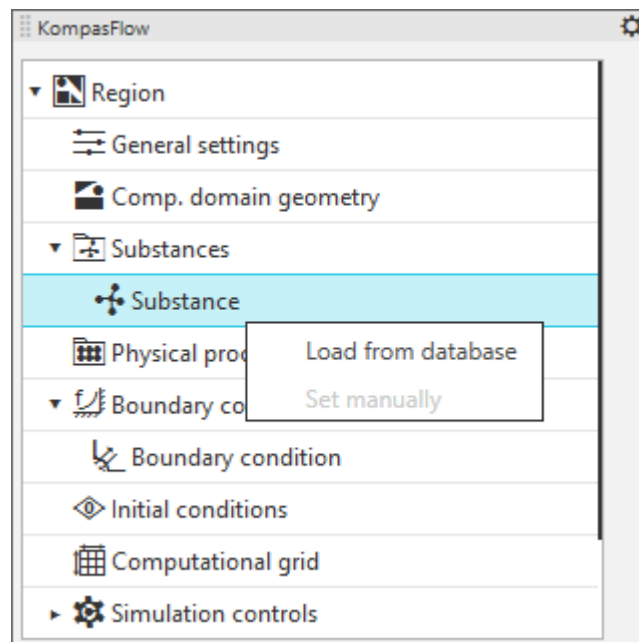
Parameters of the simulated substance are set in the property bar of the **Region > Substance** element in the KompasFlow project tree:




In our training case water flow will be simulated.

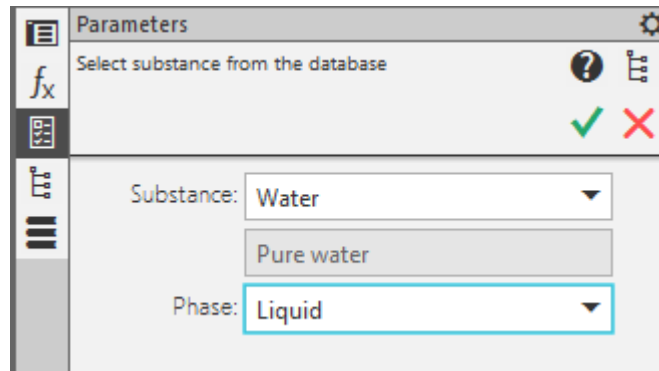
1. Open the context menu of the Substance element and select "Load from database"

Right-click the element Substance in the KompasFlow project tree. The context menu will open; select there Load from database:




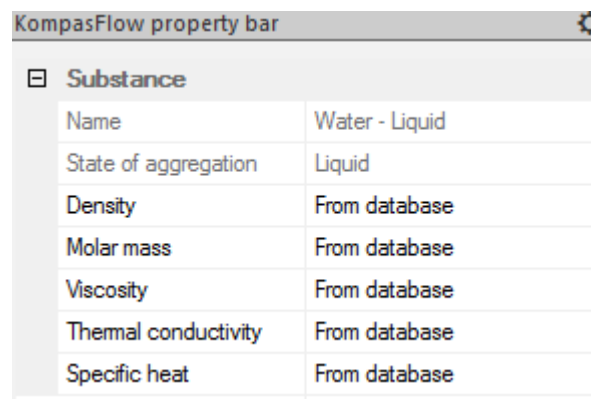
2. Selecting the Substance and its Phase

In the KOMPAS-3D control panels area, in the Parameters tab select the Substance and its Phase. Set Substance = Water and Phase = Liquid (select values from drop-down lists). When these parameters are set, the  icon will appear:



3. Confirmation of the Substance selection

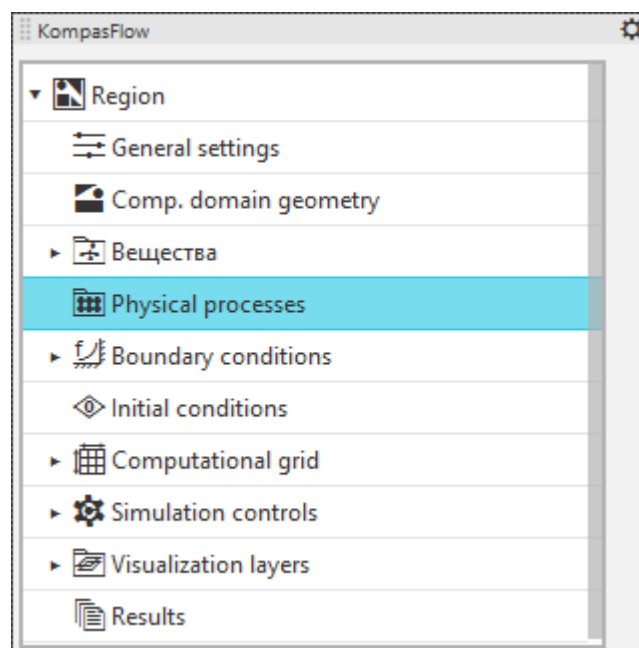
Click the  icon. The KompasFlow project tree will open again, the name of the substance and phase will be indicated in the properties of Substance, and values of physical property will be specified as From database:



KompasFlow property bar	
Substance	
Name	Water - Liquid
State of aggregation	Liquid
Density	From database
Molar mass	From database
Viscosity	From database
Thermal conductivity	From database
Specific heat	From database

4.5 Physical processes

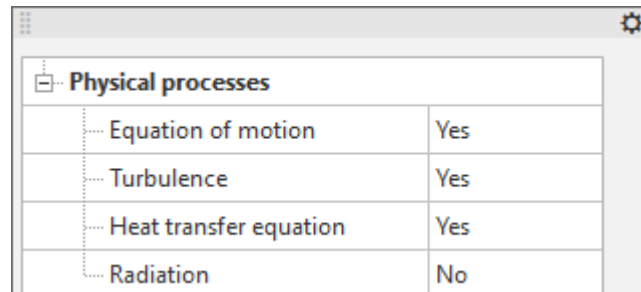
At this step, you have to specify, which physical processes will be simulated.



The set of solvable equations and their settings are specified in the properties of the Region > Physical processes element of the KompasFlow project tree:

Set the following values for Physical processes:

Equation of motion	Yes
Turbulence	Yes
Heat transfer equation	Yes
Radiation	No



4.6 Boundary conditions

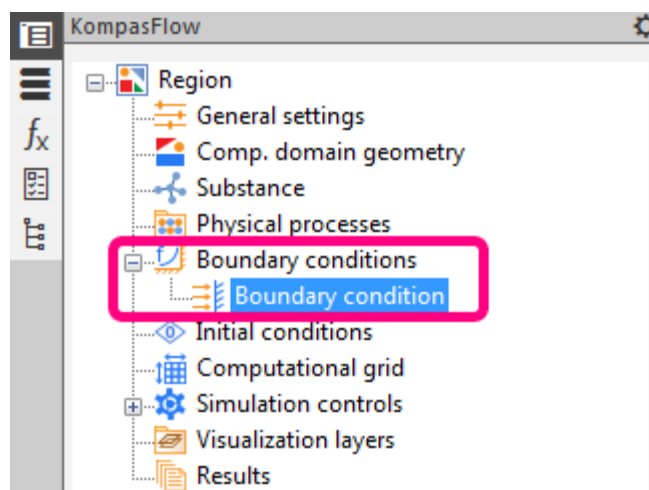
Simulation of processes in infinite space is impossible; therefore it is limited to some computational domain, on the boundaries of which it is necessary to adjust the boundary conditions to adequately describe the space outside the computational domain (see details and interface description in the [Boundary conditions](#)).

The following Boundary conditions (BCs) are implemented in KompasFlow:

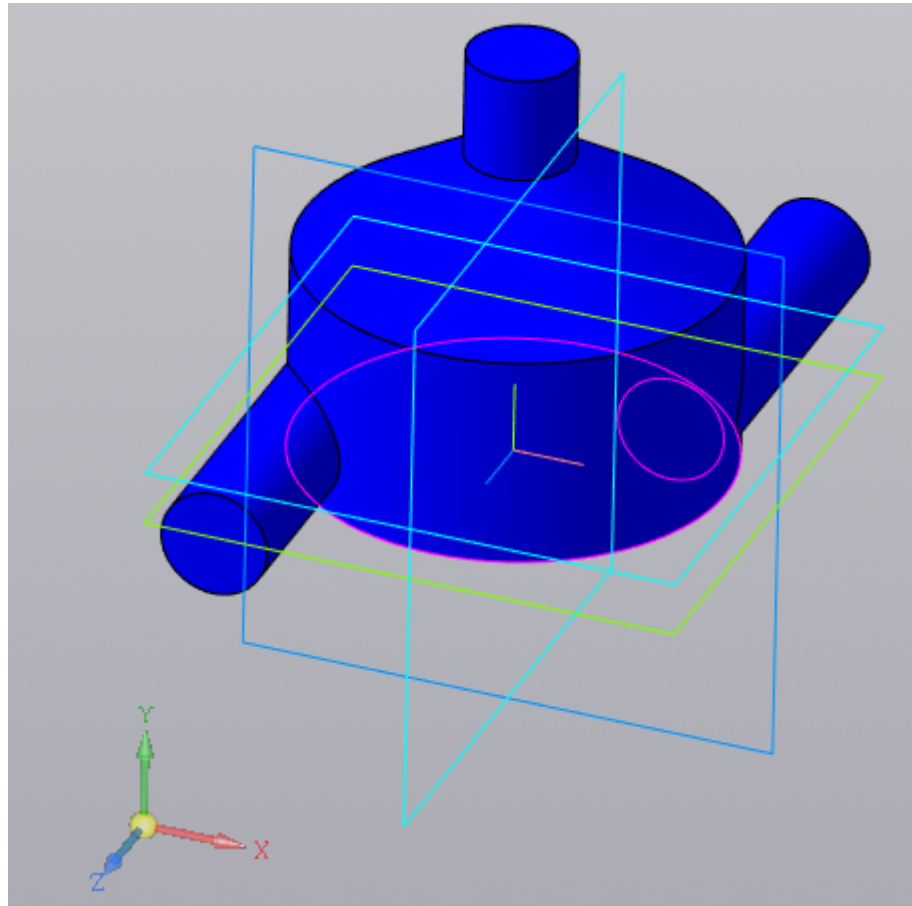
- [Symmetry](#)
- [Wall](#)
- [Inlet/Outlet](#)
- [Free outlet](#)
- [Non-reflecting](#)

Boundary conditions are specified on the faces of the model selected in the graphical area (see the step-by-step procedure in the subsection Assigning boundary conditions to groups of facets below).

Initially on all surfaces of the geometrical model a boundary condition of the Wall type with name Boundary condition is set:



This boundary condition is set on all the faces of the model and, since displaying in blue is set in its properties, the whole image of the mixer in the graphical area of KOMPAS-3D will be blue:



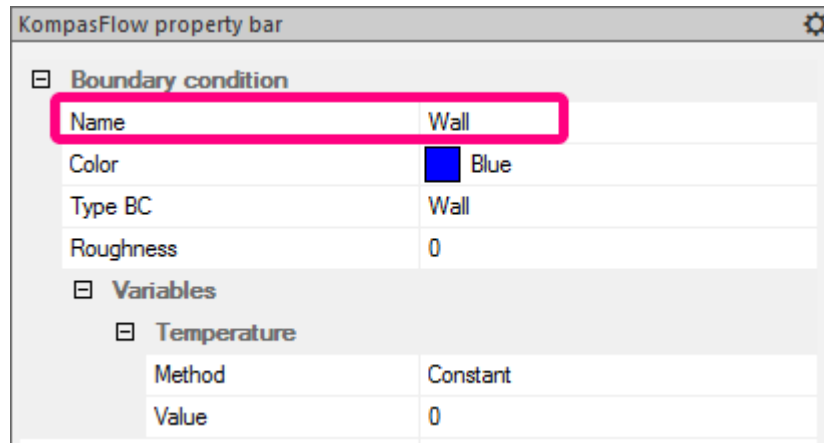
To set Boundary conditions (BCs) in our training case we have to:

1. Create four Boundary conditions: for inlet of cold water, for inlet of hot water, for outlet of mixed water and for walls.
2. Link the new BCs and corresponding faces of the computational domain.
3. Set temperature and rate values for boundary conditions that correspond to inlets of cold and hot water.

Creating boundary conditions

1. Rename existing Boundary condition

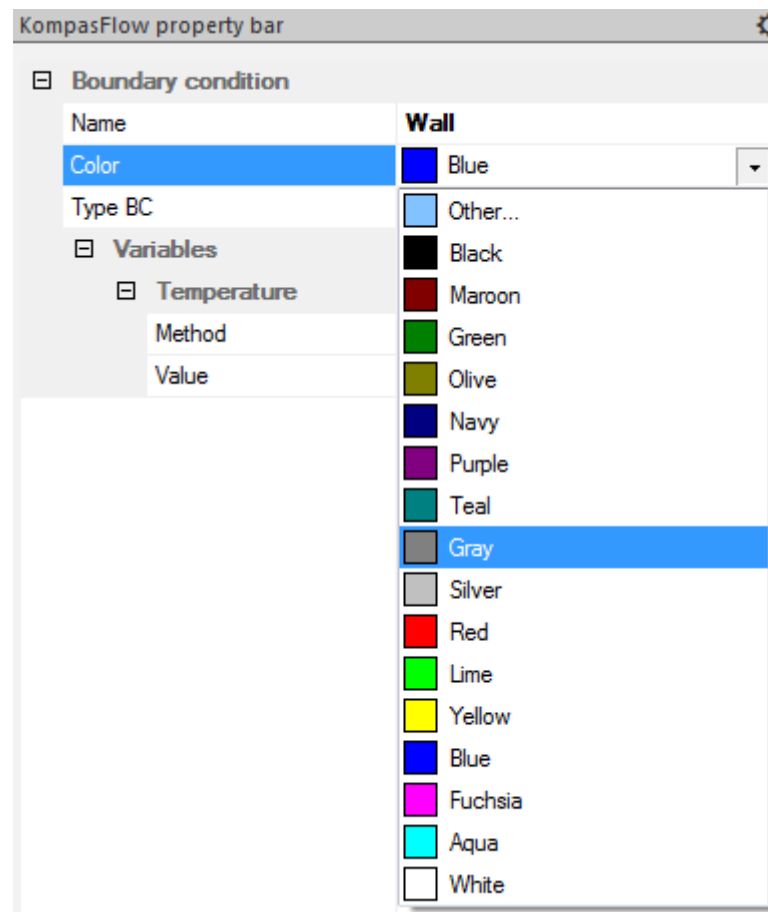
In properties of the project tree's element **Region > Boundary conditions > Boundary condition**, set **Name** as **Wall**:



This boundary condition will be used for the walls of the mixer, its name in the project tree will change from **Boundary condition** to **Wall**.

2. Change the color of the Boundary condition

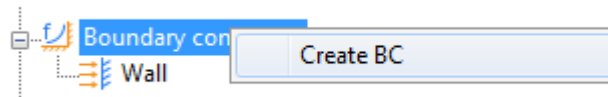
Click **Region > Boundary conditions > Wall** and in properties window set **Color = Gray** (select the color from the list):



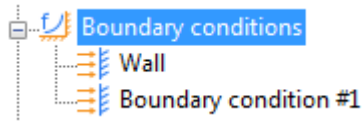
As a result, all the faces, to which this BC is assigned, will change their color.

3. Create another Boundary condition (for inlet of the cold water)

Open the context menu of **Region > Boundary conditions** and select **Create BC**:



Boundary condition #1 item appears in **Boundary conditions** folder:



4. Change the name and type of newly created Boundary condition #1

Select the element Boundary condition #1 in the project tree and set in its properties:

Name	Cold inlet
Color	■ Blue (by default)
Type BC	Inlet/Outlet

Step 5. Create one more Boundary condition (for inlet of the hot water), set its name, color, and type.

Open the context menu of the folder Region > Boundary conditions and select command Create. In the Boundary conditions folder the Boundary condition #1 element will appear. Set its properties:

Name	Hot inlet
Color	■ Red
Type BC	Inlet/Outlet

Step 6. Create one more Boundary condition (for outlet), set its name, color, and type.

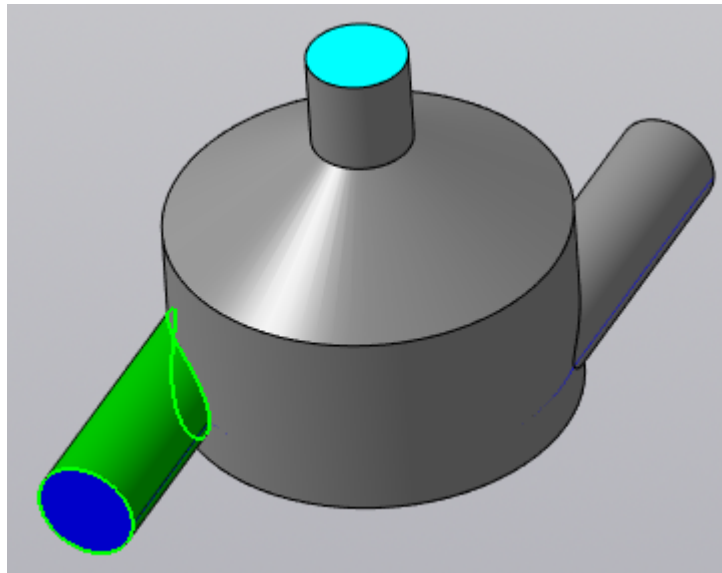
Open the context menu of the folder Region > Boundary conditions and select command Create. In the Boundary conditions folder the Boundary condition #1 element will appear. Set its properties:

Name	Outlet
Color	■ Aqua
Type BC	Free outlet

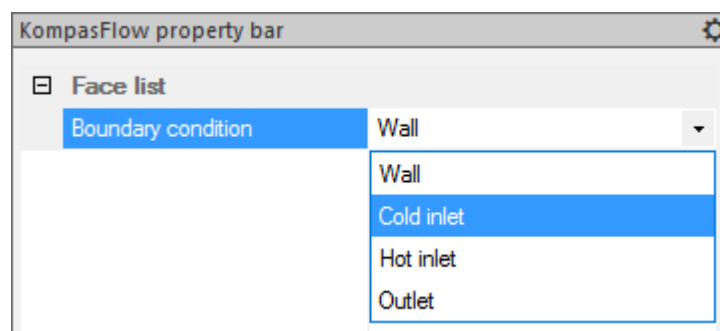
Assigning boundary conditions to groups of facets

We have to set **Boundary conditions** on the groups of facets (faces of the geometrical model) that correspond to inlets of hot and cold water, outlet of mixed water and walls of the mixer.

To select the desirable group of facets, click the image of the corresponding surface in the graphical area of KOMPAS-3D (after this the image will be highlighted green):

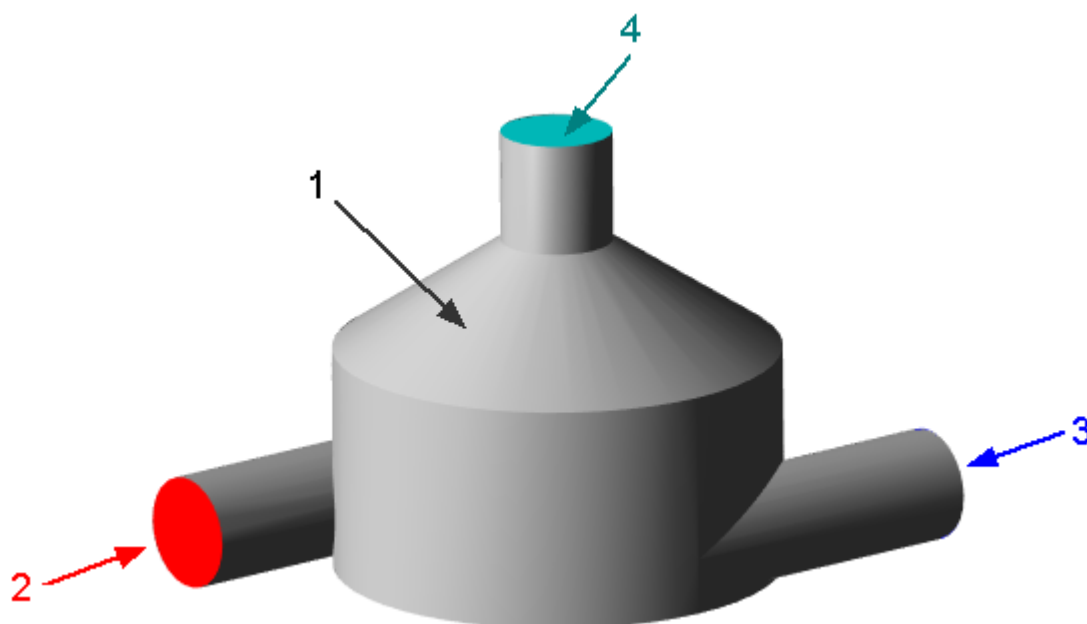


In the property bar, you will see a root element **Face** list that contains the **Boundary condition** parameter with a value, which is selected from a drop-down list:



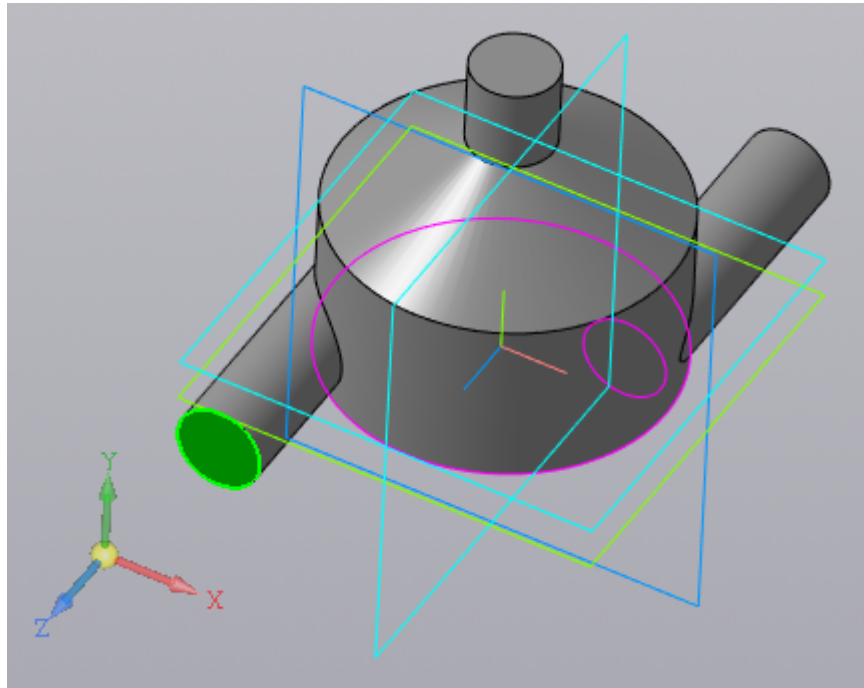
Let us assign the created boundary conditions to faces of the geometrical model.

Since the boundary condition **Wall** is already assigned to all groups of facets, we just need to change boundary conditions on cross-sections of inlet tubes and the outlet tube according to the illustration below:



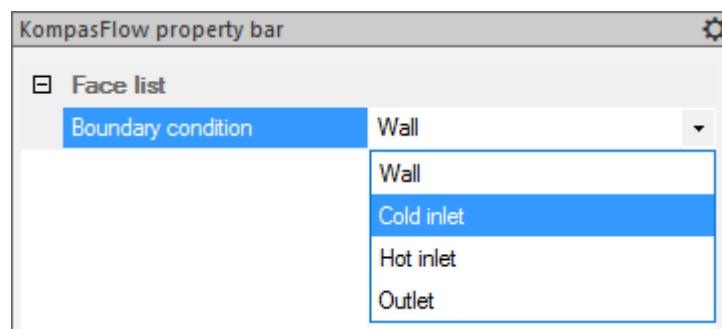
1 – Wall, 2 – Hot inlet, 3 – Cold inlet, 4 – Outlet

Click the image of the cross-section of an inlet tube in the graphical area of KOMPAS-3D, and it will be highlighted green.



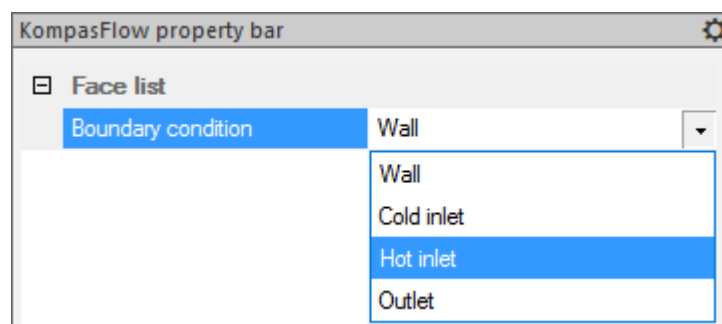
In property bar, parameters of the corresponding element **Face** list will open.

Set **Boundary condition** = **Cold inlet**:



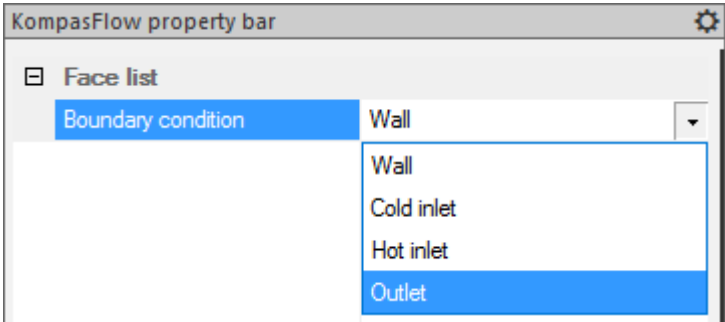
Rotate the image of the mixer in the graphical area and click the cross-section of the other inlet tube.

In the property bar, parameters of corresponding element Face list will open; set there Boundary condition = Hot inlet:



If the cross-section of outlet tube is not visible, rotate the image of the mixer so that it would be visible. Click the cross-section of the outlet tube.

In the property bar, parameters of corresponding element Face list will open; set there Boundary condition = Outlet:



After assigning the boundary conditions, the corresponding sections of tubes will displayed in the graphical area colored in blue, red and light blue colors.

Setting parameters of boundary conditions

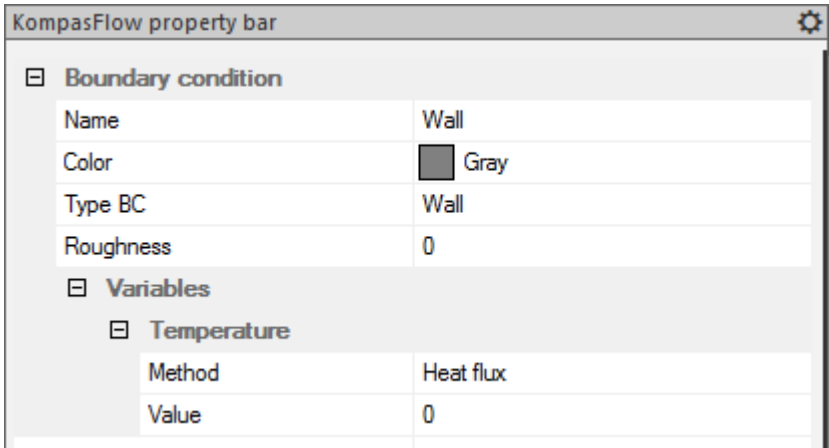
Now we have to define methods of how values of physical quantities will be calculated on the boundaries of the computational domain.

Select Boundary conditions one by one and set their parameters in the property bar.

Set parameters for **Wall** boundary condition:

Variables	
Temperature	
Method	Heat flux
Value	0

These parameters mean that the wall of the mixer is impermeable for heat transfer. These changes will be saved in the property bar of Wall:



Set parameters for **Cold inlet** boundary condition:

Variables	
Velocity	
Method	Normal mass velocity

Value	1273.24
Temperature	
Value	5

Here the normal mass flow rate is set, $[\text{kg}/(\text{m}^2 \cdot \text{s})]$. The value of $1273.24 \text{ kg} \cdot \text{m}^{-2} \cdot \text{s}^{-1}$ is obtained by dividing mass flow rate $0.1 \text{ kg} \cdot \text{s}^{-1}$ by area of the cross-section of the inlet tube of diameter 0.01 m ($7.854 \times 10^{-5} \text{ m}^2$).

The temperature value is 5 [K] higher than the reference temperature value of 273 [K] , i.e. it corresponds to a temperature of $5 \text{ }^\circ\text{C}$.

Set parameters for **Hot inlet** boundary condition set a similar **Velocity** value and an increased **Temperature** value:

Variables	
Velocity	
Method	Normal mass velocity
Value	1273.24
Temperature	
Value	70

The temperature value is 70 [K] higher than the reference temperature value of 273 [K] , i.e. it corresponds to a temperature of $70 \text{ }^\circ\text{C}$.

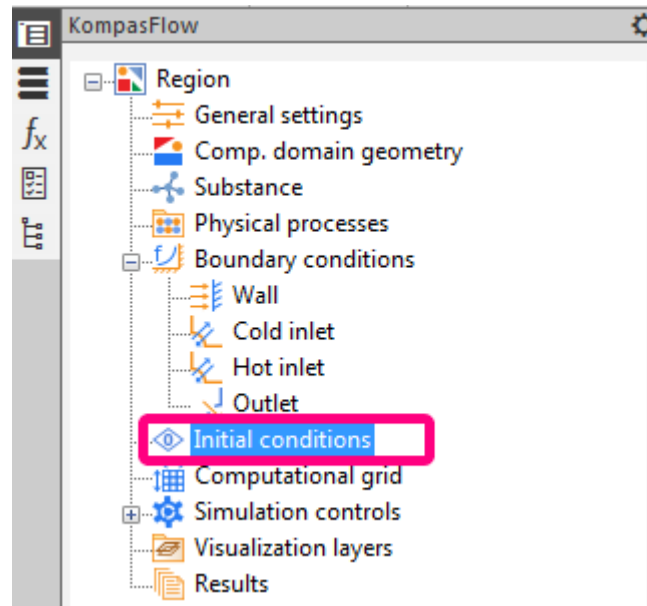
Set parameters for **Outlet** boundary condition:

Variables	
Velocity	
Pressure	0
Temperature	
Value	5

The pressure value is set equal to the reference pressure, specified in $[\text{Pa}]$.

The temperature value is set to $5 \text{ }^\circ\text{C}$. This temperature is used exclusively for reverse flow through the boundary condition.

4.7 Initial conditions



The Initial conditions are used to specify values of the simulated variables in the computational domain at the initial time moment.

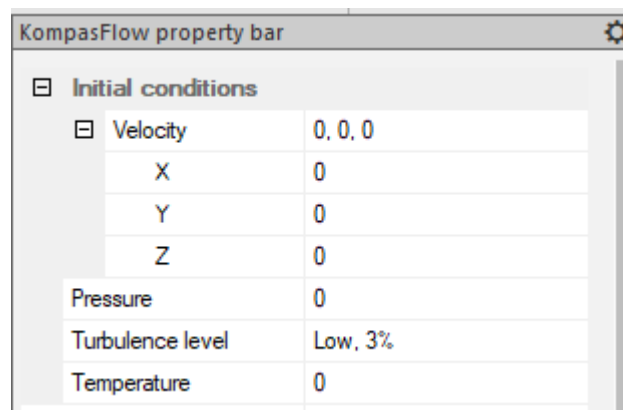
By default all numerical variables at the initial time moment have zero values and **Turbulence level = Low, 3%**:

Setting nonzero initial conditions allows you to speed up the convergence of the solution (for example, in tasks of external flow, it is convenient to set the initial velocity field to the velocity of the forward flow).

In this case, Initial conditions correspond to fluid with temperature 5°C (i.e. 5 [K] higher than the [reference temperature](#) 273 [K]) and pressure equal to the [reference pressure](#) (101000 [Pa]).

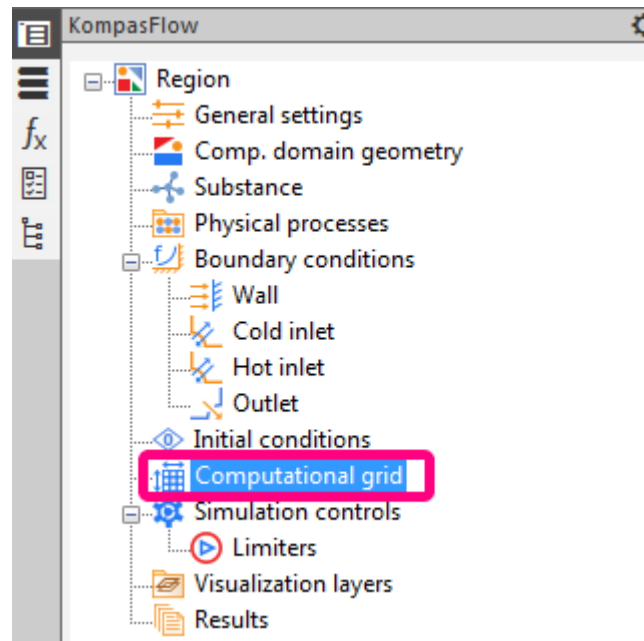
Set **Initial conditions** in the property bar:

Temperature	5
-------------	---



Do not change default values of parameters **Velocity (X, Y, Z)** and **Pressure**.

4.8 Computational grid



It is impossible to calculate movement of each molecule. That is why discretization of space with the computational grid is used. The whole calculated volume is divided into elementary volumes (cells). The smaller the cells are, the better is the space resolution. In the volume of each cell, all physical variables are considered constant at the current time moment.

In our training case, specifying the Computational grid consists of two stages:

- a) specifying the [initial Computational grid](#)
- b) specifying a local [Adaptation](#)

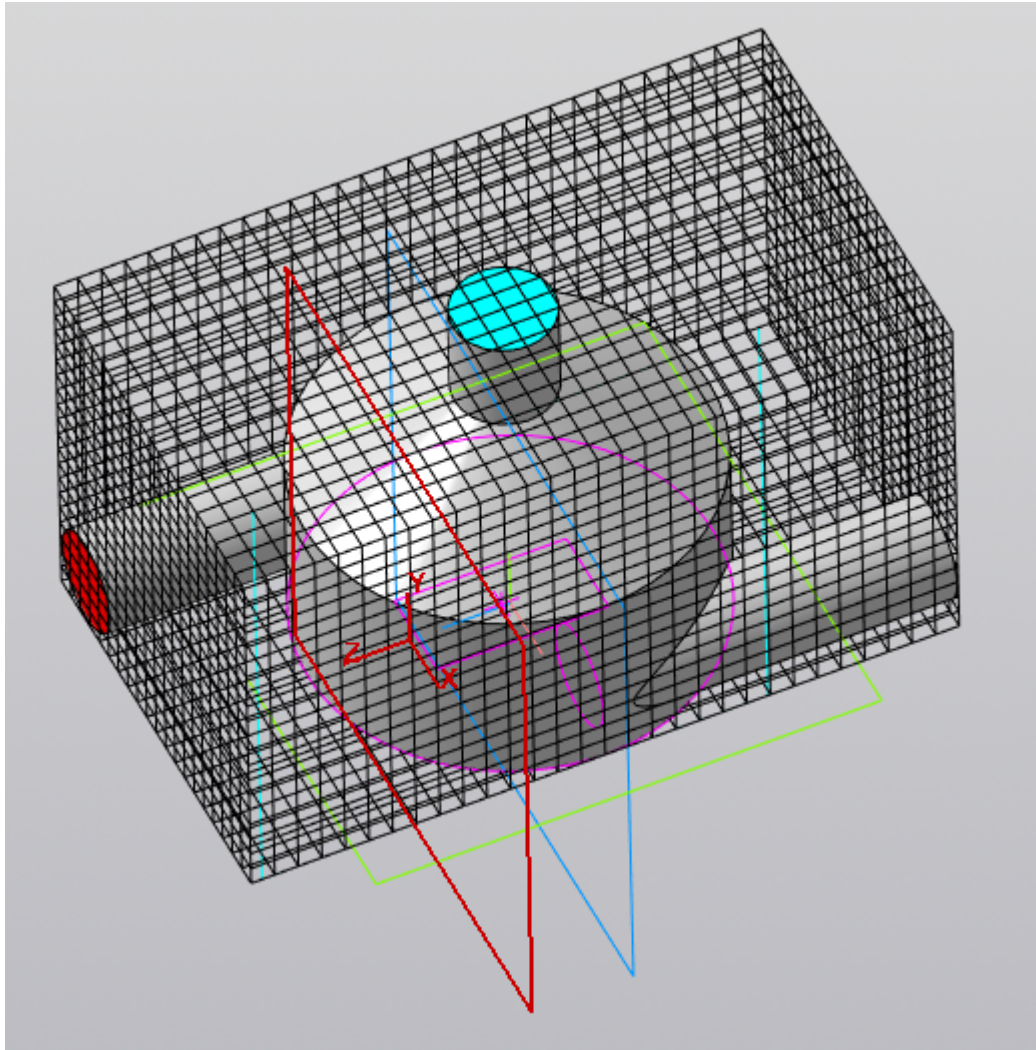
4.8.1 Initial computational grid

The initial computational grid in KompasFlow is orthogonal and uniform. It is set in the box describing the computational domain. You need to provide steps for its division into 3 coordinate axes.

Set the uniform initial computational grid with the number of cells 20, 20, and 30 along axes X, Y, and Z respectively, and specify the following properties of the [Region > Computational grid](#) element:

nX	20
nY	20
nZ	30

The image of the Computational grid in the graphical area will change accordingly:



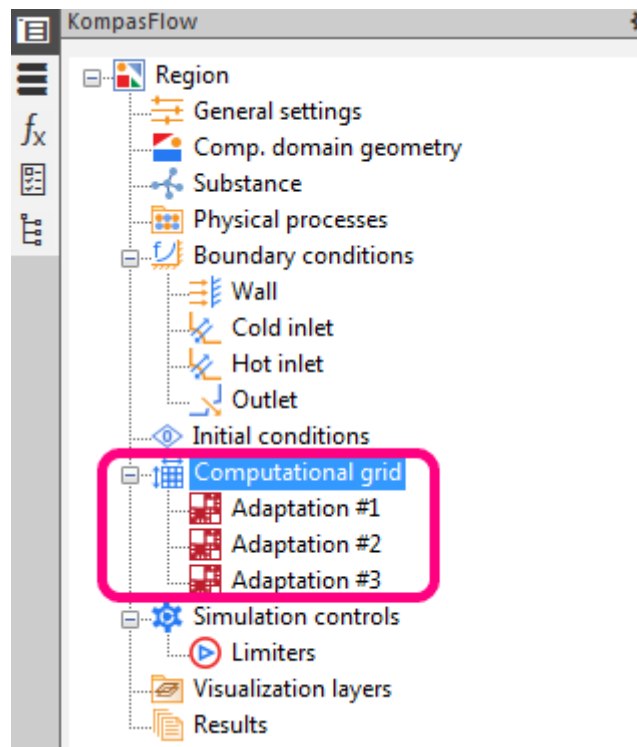
4.8.2 Adaptation

Adaptation is a method of refinement the initial grid near a surface of the geometrical model. Adaptation causes refinement the cells by dividing the grid twice along directions of each coordinate axis (X, Y, Z), so one partitioning splits a cell into 8 parts.

Adaption level is the number of consecutive partitioning of the grid cells. A higher the level is, the smaller the cells are.

It is possible to specify how many adaptation layers will be formed by adaptations of each level. This parameter allows you to expand adaptation at some distance from the surface, on which it is set (in direction along the normal to this surface).

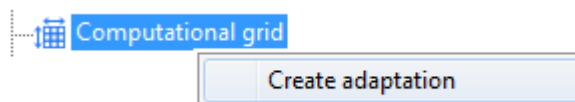
Adaptation is specified in elements [Region > Computational grid > Adaptation #N](#) in the KompasFlow project tree:



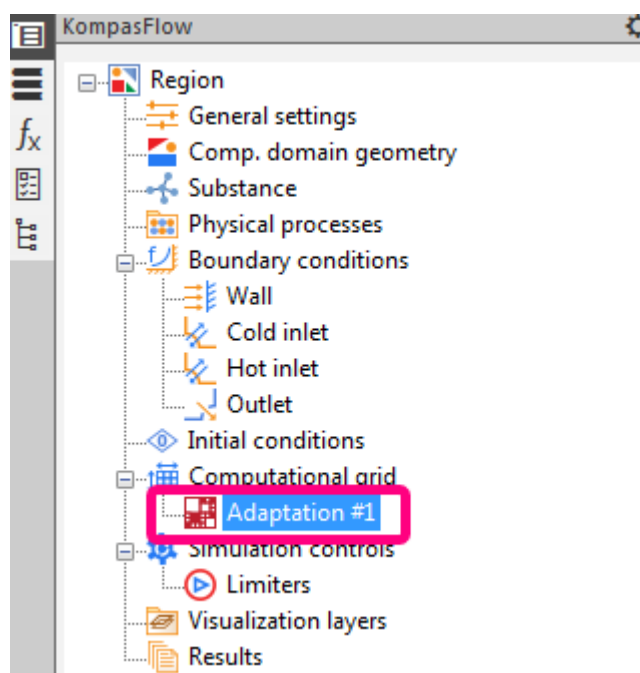
In our training case, we specify an adaptation on surfaces of the mixer's walls, with partitioning of no more than one level, with three layers of adapted cells adjacent the walls of the mixer.

Proceed as follows:

Create an Adaptation that will be used on walls of the mixer. To do this, in the context menu of [Region > Computational grid](#), select Create adaptation:

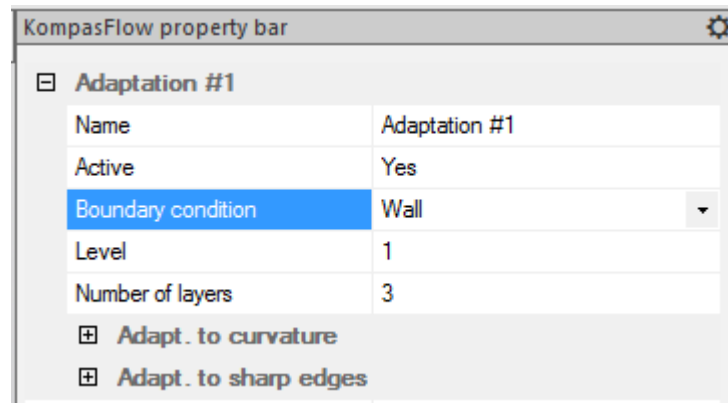


In the project tree the Adaptation #1 element will appear:



Set the following parameters of Adaptation #1 in its property bar:

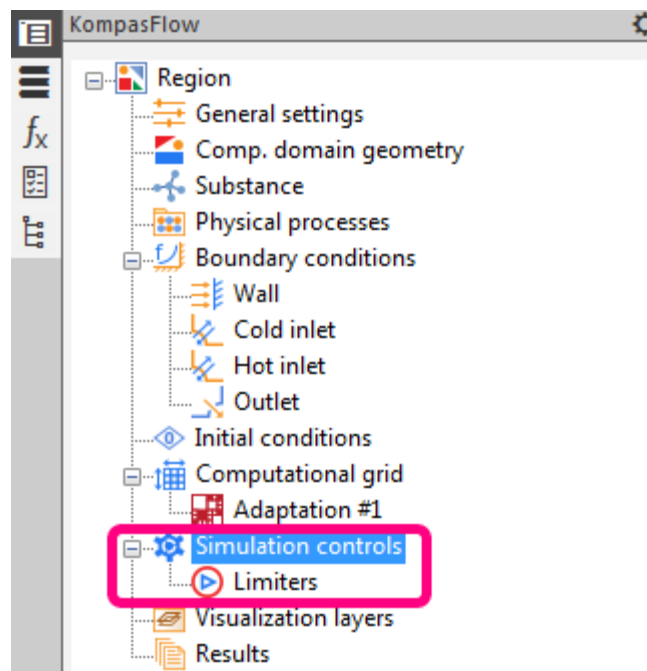
Active	Yes
Boundary condition	Wall
Level	1
Number of layers	3



Do not change default parameters in **Adapt. to curvature** and **Adapt. to sharp edges** groups.

4.9 Parameters of calculation controls

Parameters of the computation are set in property bars of **Region > Simulation controls** and **Region > Simulation controls > Limiters**:

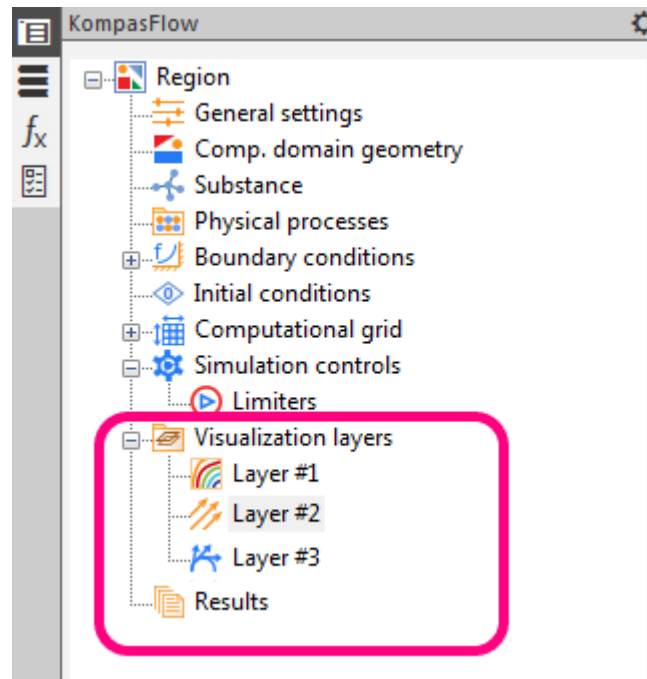


Set the following parameters in the property bar of **Simulation controls** item:

CFL	50
Max time step	0.01
Simulated time	15

Do not change default settings of parameters of **Simulation controls > Limiters**.

4.10 Viewing results of the computation



To set displaying results of the computation, the following folders of the project tree are used:

- **Visualization layers** (it contains elements Layer #N)
- **Results** (it contains elements Result #N)

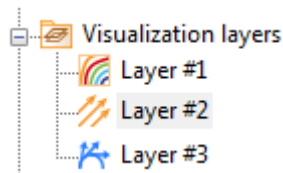
Images of the Layers are displayed in the graphical area of KOMPAS-3D (see [Viewing visualization layers during the computation](#)).

Plots with **Results** are displayed in the Monitoring window (see [Data viewing in the Monitoring window](#)).

4.10.1 Creating layers

Visualization layers are used to visualize results of the computation.

Each layer is presented in the project tree as a **Layer #N** child item in the **Visualization layers** folder:

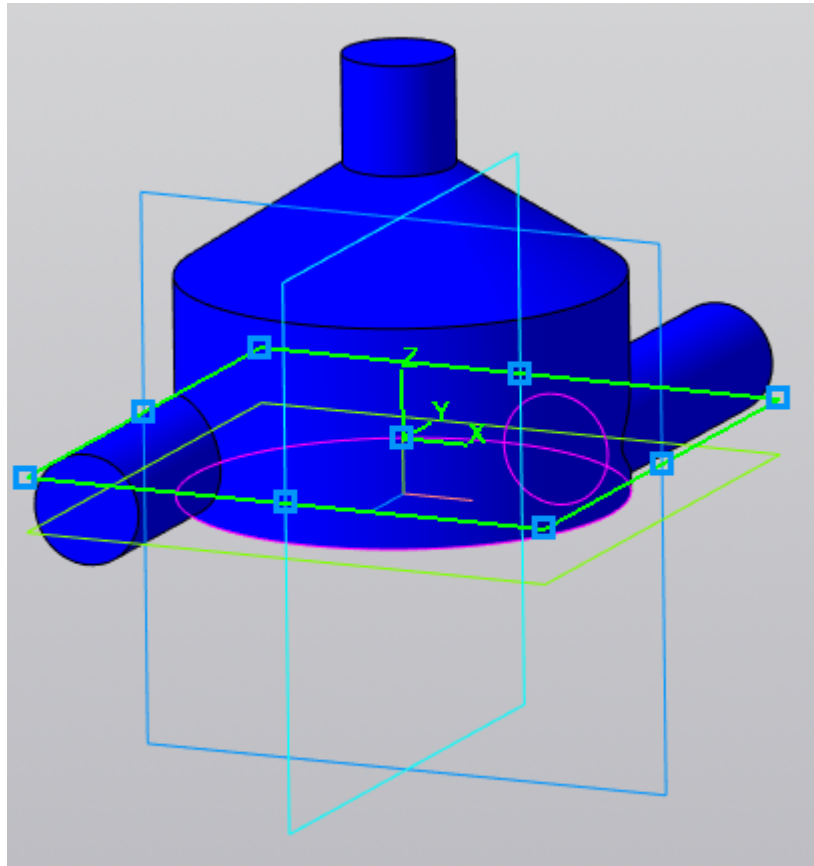
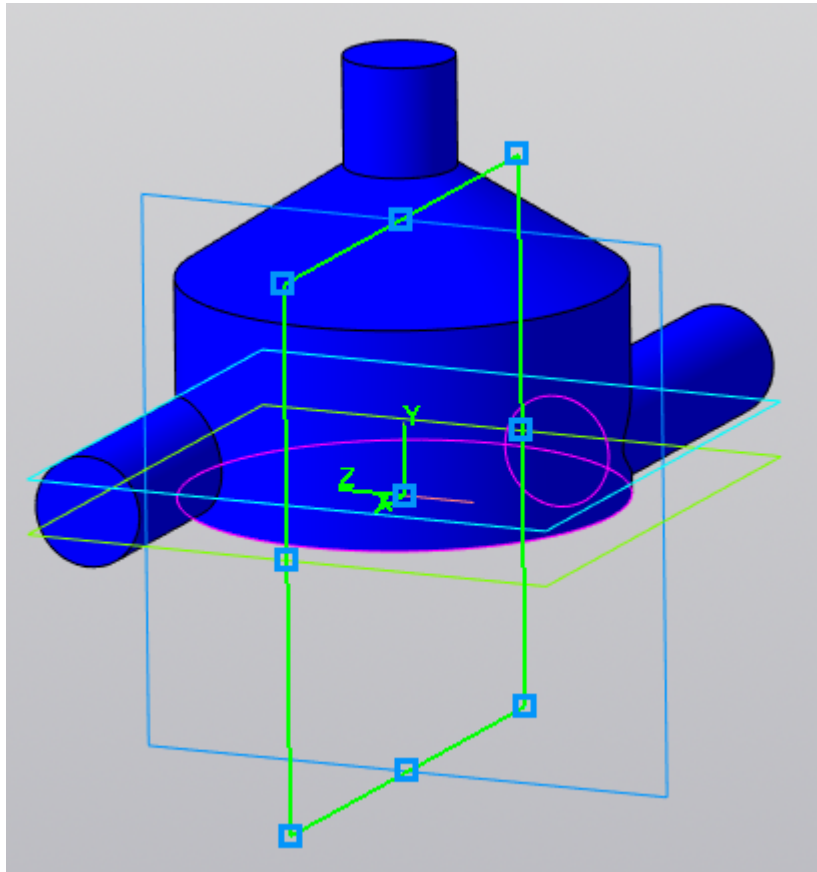


Layers are built in space, or on planes, or on geometrical surfaces.

In our training case, we will built Layers built on the following Planes:

- one of the Planes that is a vertical plane parallel to inlet tubes and intersecting the center line of the mixer
- the other Plane is a horizontal plane intersecting the center lines of inlet tubes

The loaded project already has these planes. They are **Offset plane:1** and **Offset plane:2**.

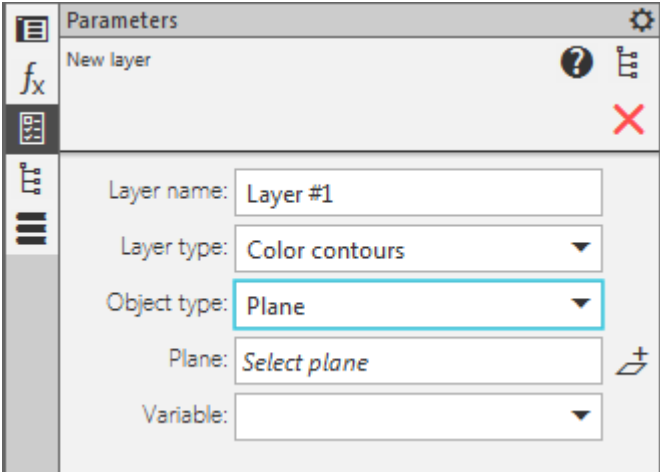


Offset planes 1 and 2 initially set in mixerfv_enu.m3d file (designated in green with 9 points)



On each of these Planes we will create two Layers, and one of them will display Temperature as colored fill, and the other will display Velocity as vectors. So, four Layers will be created. If you wish, you can create in KOMPAS-3D more Planes and build Layers on them.

Proceed as follows:


1. In the context menu of Visualization layers element, select the command Create Layer. In the KOMPAS-3D control panels the Parameters tab will open where you have to set parameters of the new layer:



2. Set parameters of **Layer #1** in **Parameters** tab:

Layer name	Layer #1
Layer type	Color contours
Object type	Plane
Plane	Offset plane:1 To specify this, click the  icon and select Offset plane:1 from the geometrical elements tree that will be displayed in a separate tab Tree and then return to the Parameters tab. The selected Plane will be presented in the tree of geometrical elements with a red icon  .
Variable	Temperature

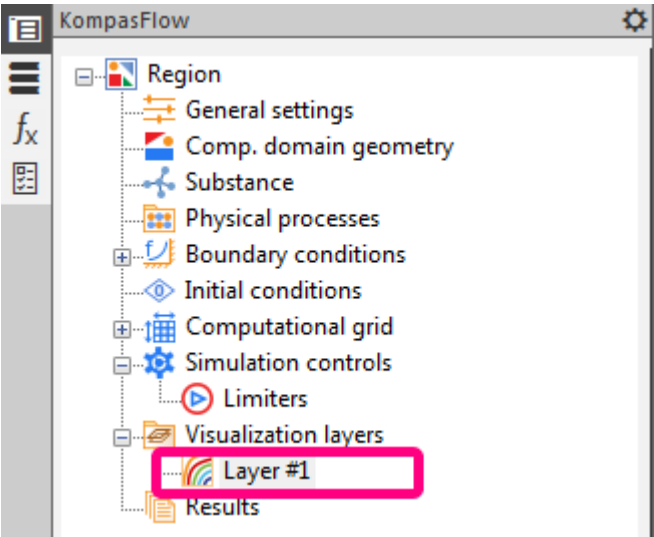
Some fields already contain the required default values. In these cases, make sure that the necessary values are set there.



Optionally parameters **Layer name**, **Plane**, **Surface** and **Variable** can be changed after creating a **Layer** (in the **Layer's** property bar).

Click  icon.

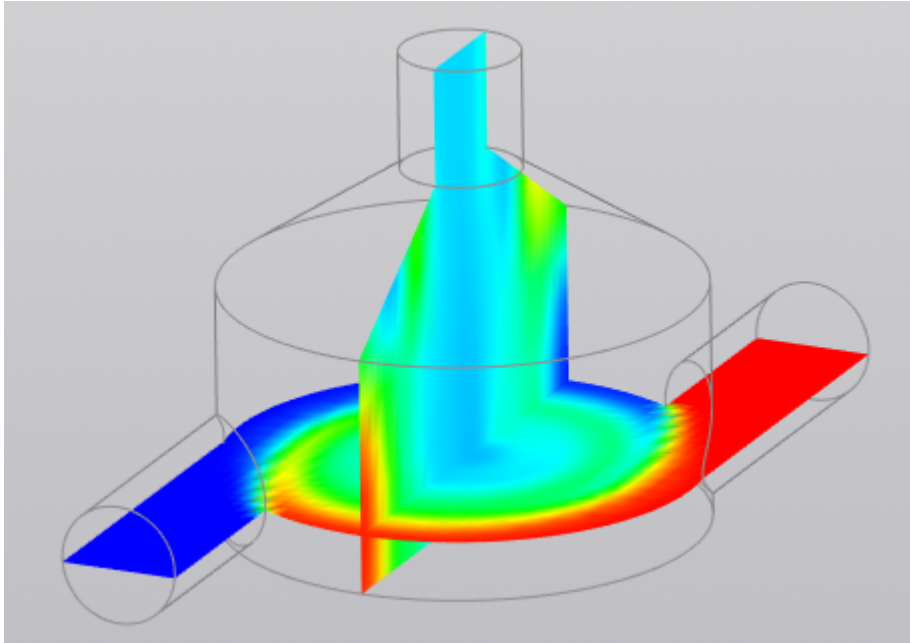
Layer #1 will be created and appears in KompasFlow project window:



4. Create in a similar way three more Layers with the following parameters:
- **Layer #2** on **Offset plane:2** with parameters **Layer type = Vectors**, **Variable = Temperature**.
 - **Layer #3** on **Offset plane:1** with parameters **Layer type = Vectors**, **Variable = Velocity**, **Color variable = No**.
 - **Layer #4** on **Offset plane:2** with parameters **Layer type = Vectors**, **Variable = Velocity**, **Color variable = No**.
5. Set the range for layers that display **Temperature**. In the properties bars of **Layer #1** and **Layer #2**, set:

Layer settings	
View settings	
Range	
Mode	Manual
Maximum	70
Minimum	5

For both layers that visualize distribution of **Temperature**, the manual display temperature range (from 5 to 70 degrees) is set. This is done so that temperature on both of these layers be displayed using the same, common to them, scale.

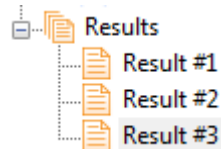


Displaying temperature distribution on two intersecting planes in a common scale (from 0 to 70 degrees)

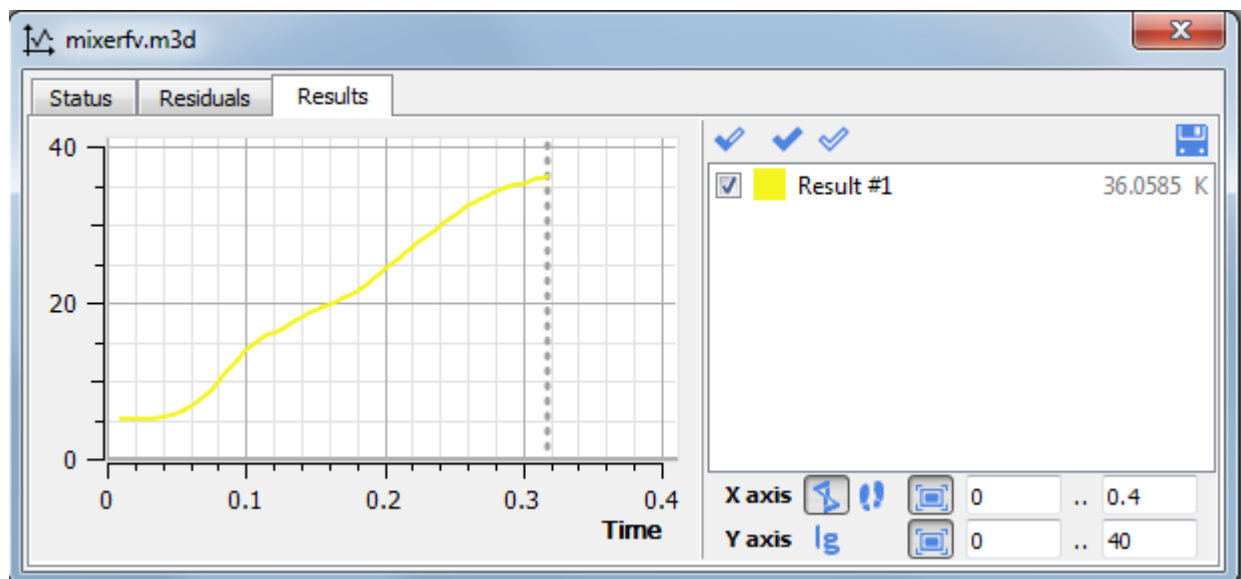
4.10.2 Creating results

Result #N items are used to calculate integral values.

They are located in the **Results** folder in the project tree:



After starting the computation, plots of Results depending on time are displayed in the [Monitoring window](#):

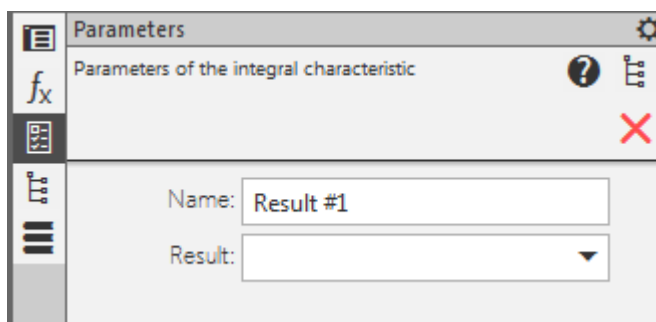


It is convenient to use plots of Results to estimate the convergence of the solution. For example, when average temperature on the mixer's outlet stop changing, you can accept that the solution is convergent and the further calculation is not required.

We will create Result #1 that calculates the average temperature of the water in the outlet of the mixer.

Proceed as follows:

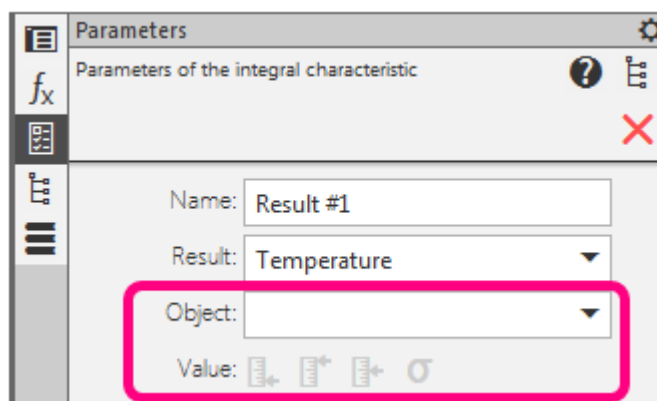
1. In the context menu of Results, select the **Create Result** command. In the KOMPAS-3D control panels, the **Parameters** tab will open where you can set parameters of the new **Result**:



2. In the Result field, select Temperature.

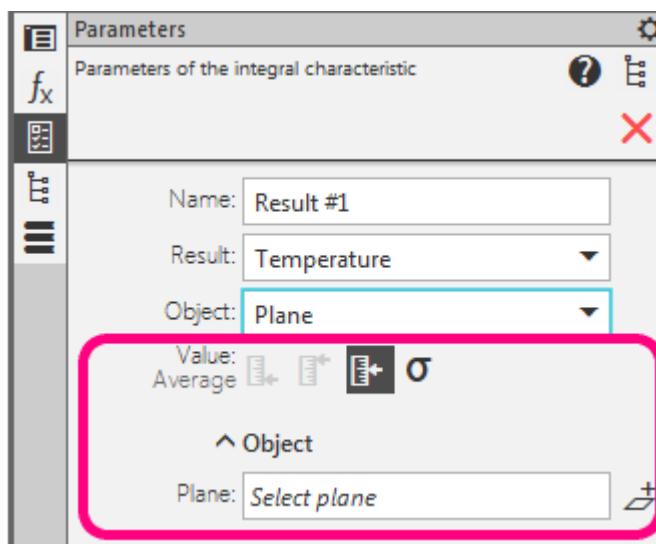
In the Parameters tab, the following fields will appear:


- Object (available options: Comp. space | Plane | Surface | Point)
- Value (at this step, this field is inactive)



3. In the Object field, select Plane.



The Value field will become active (available options: Average | Std.dev.; by default Average is selected) and the Plane field will appear:




4. Do not change the selection **Value = Average** () and click the  icon on the right of the **Plane** field to create a new **Plane**.

Result #1 will be calculated as the average temperature at the cross-section of the outlet by a new **Plane** displaced relating to the boundary condition **Outlet**.

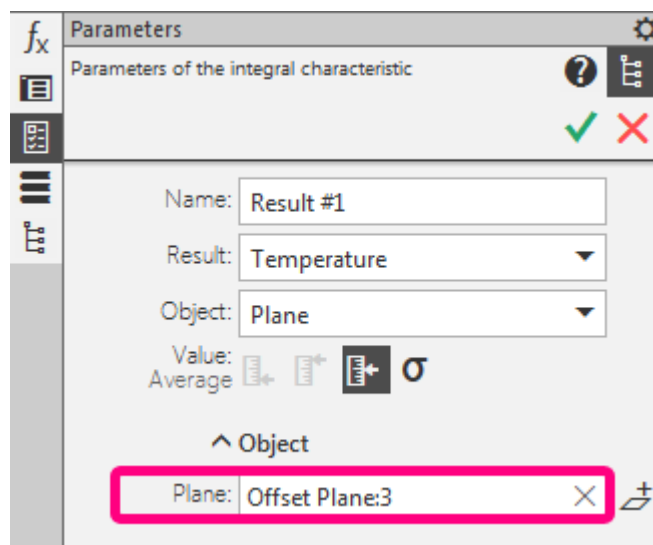
5. In the Parameters tab, a form will open where you have to create a new Offset plane.

Click the , icon, then click the object of origin of coordinates () to expand it and select plane ZX.

Create an offset plane that is parallel to the XZ plane displaced at 36 mm from it.

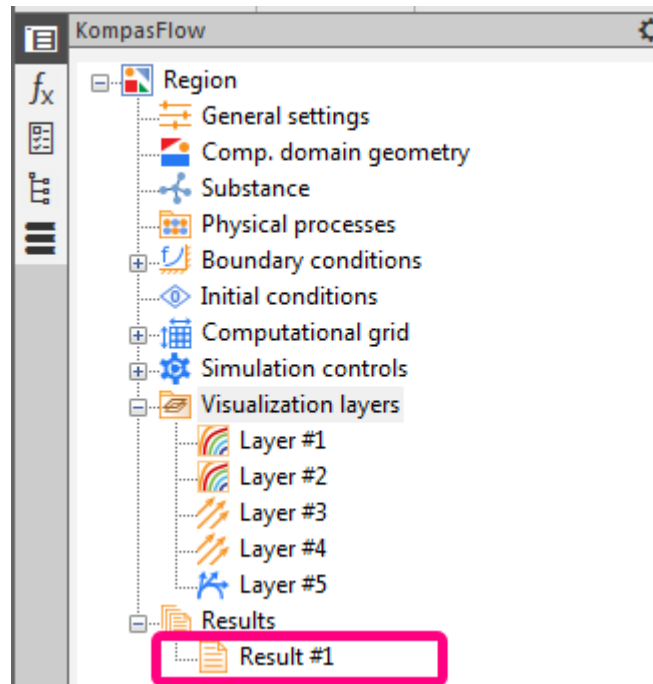
Then click the  icon.

6. You will return to the previous form for setting parameters where the Plane field will be filled as Offset plane:3:



7. Click the  icon.

KompasFlow tab will be opened and Result #1 appears:



This step-by-step procedure describes creation of a **Result** in the cross-section of the computational domain by a specially created **Offset plane**.

If the geometrical model already contains surfaces suitable for calculating a **Result**, you can set **Object = Surface** and then select the desired surface from the graphical area of KOMPAS-3D.

After starting the project's computation, you can view in the [Monitoring window](#) a plot of changing average temperature of water at the outlet of the mixer.

4.10.3 Starting the project's computation

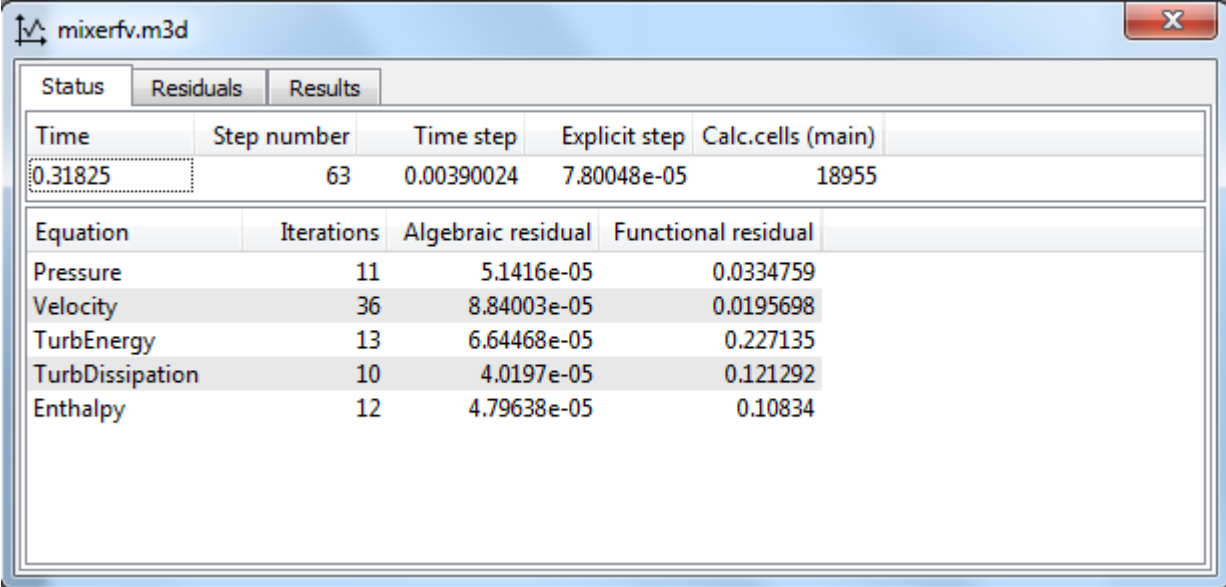
Start computation of the project:

In the [KompasFlow toolbar](#) click the  Start solve icon.




After installing KompasFlow, its first running for calculation can cause a delay of more than a minute.



The computation will start, and the [Monitoring window](#) will open:




Time	Step number	Time step	Explicit step	Calc.cells (main)
0.31825	63	0.00390024	7.80048e-05	18955

Equation	Iterations	Algebraic residual	Functional residual
Pressure	11	5.1416e-05	0.0334759
Velocity	36	8.84003e-05	0.0195698
TurbEnergy	13	6.64468e-05	0.227135
TurbDissipation	10	4.0197e-05	0.121292
Enthalpy	12	4.79638e-05	0.10834

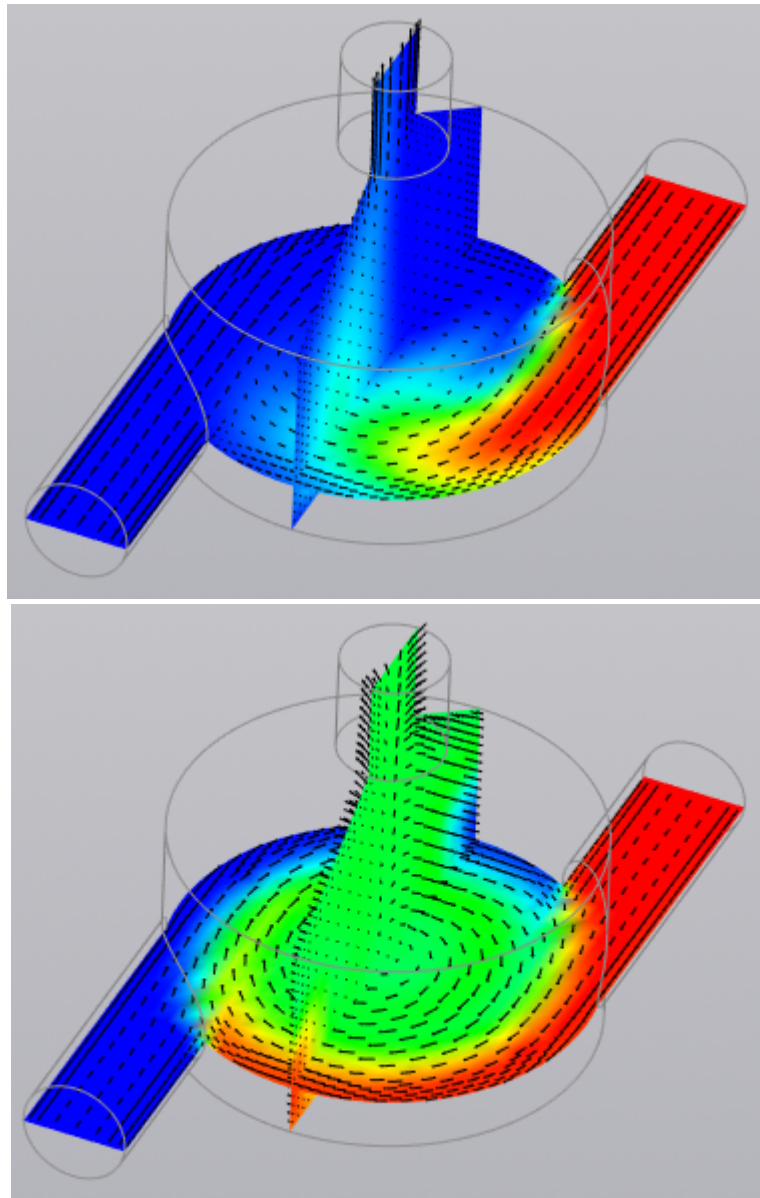
Monitoring window can be moved to a convenient place or closed by clicking the x symbol in its upper right corner. To reopen the Monitoring window, apply the  Open monitor command from the [KompasFlow toolbar](#).

Computation can be stopped by clicking the  (Stop) icon in the [KompasFlow toolbar](#) and resumed by clicking the  (Continue) icon.

4.10.4 Viewing visualization layers during the computation

If necessary (and the icon is not already pressed), click the  (Frame) icon in the graphical area to make transparent the outer surfaces of the object, that enclose the internal space.

This allows you to view the flow in the previously set Layers that have been built on mutually perpendicular Planes (Offset plane:1 and Offset plane:2):



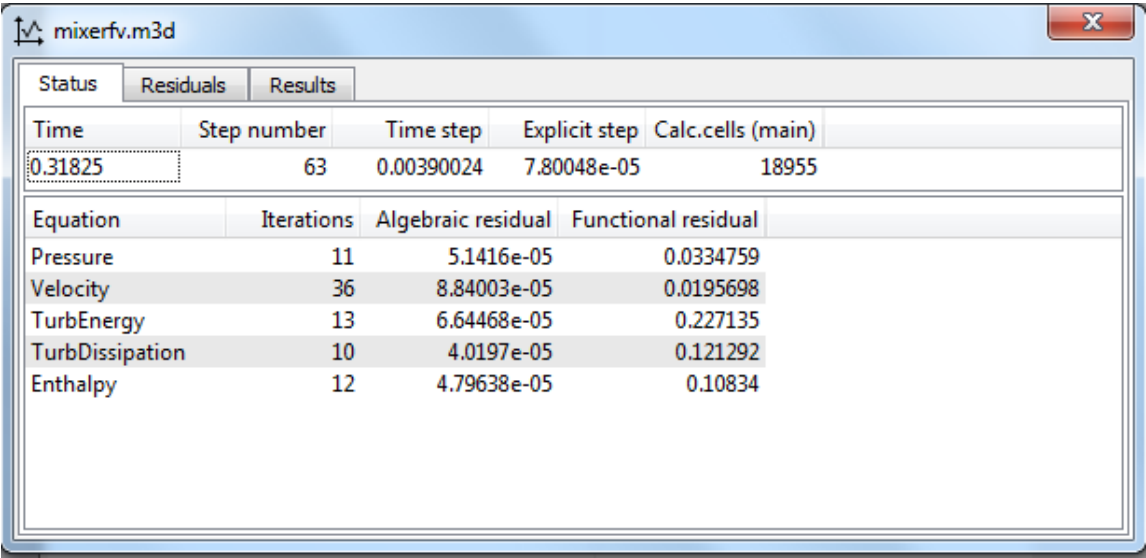
Displaying layers with temperature field and velocity vectors:
a – just after starting the calculation, b – the final result

Using KOMPAS-3D, find a convenient viewpoint to observe the flow.

4.10.5 Data viewing in the Monitoring window

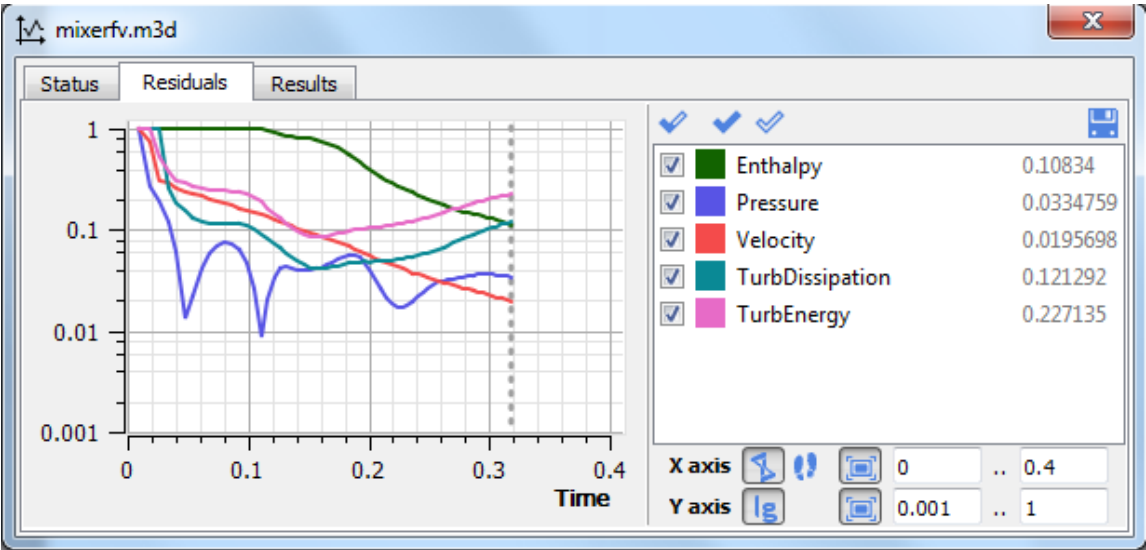
The Monitoring window has three tabs where you can observe during the calculation:

- a) The Status tab displays parameters of the computation in tabular form:

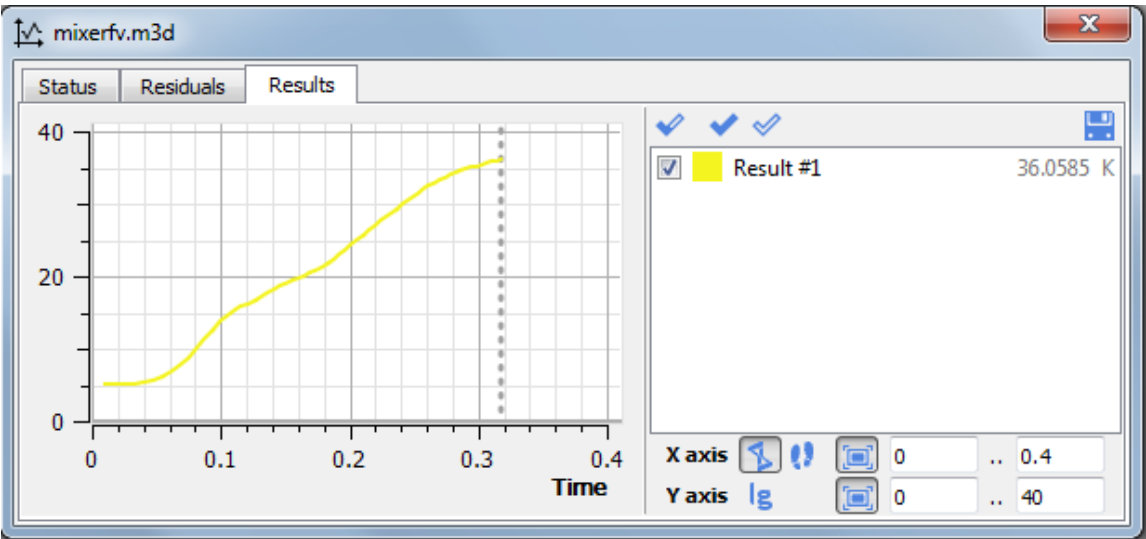


Monitoring window, the Status tab

b) Tabs Residuals and Results display plots of control parameters and plots of **Results**, which you have created before:



Monitoring window, the Residuals tab



Monitoring window, the Results tab

5 Interface reference guide

This chapter contains detailed description of KompasFlow interface.

See sections:

[KompasFlow toolbar](#)

[Wizard](#)

[Templates](#)

[Items in the project window](#)

[Region](#)

[General settings](#)

[Simulating an external flow](#)

[Computational domain geometry](#)

[Substance](#)

[Physical processes](#)

[Boundary conditions](#)

[Symmetry](#)

[Wall](#)

[Inlet/Outlet](#)

[Free outlet](#)

[Non-reflecting](#)

[External BC](#)

[Source of moisture](#)

[Hood](#)

[Initial conditions](#)

[Computational grid and adaptation](#)

[Computation parameters and limiters](#)

[Visualization layers](#)

[General properties of layers](#)

[Color contours](#)

[Vectors](#)

[Streamlines](#)

[Computation grid section](#)

Isosurface

Results (folder)

Results (items)

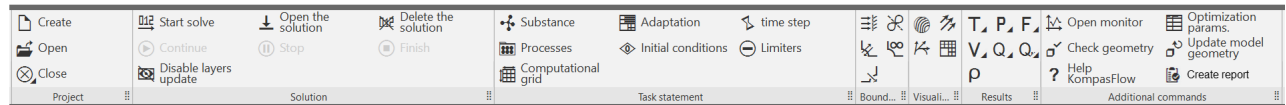
Monitoring window

Changing a project connected to the solver

Optimization

Creating reports

5.1 KompasFlow toolbar



KompasFlow toolbar contains sets of commands:

Project


To create, open, close, save and delete a KompasFlow project without affecting the geometrical model (KOMPAS-3D project).

	Create	Create a new KompasFlow project
	Open	Open an existing (previously saved) KompasFlow project
	Close	<ul style="list-style-type: none">close the current KompasFlow project (click)call options (click and hold):<ul style="list-style-type: none"> Close – close the current KompasFlow project. Delete – exclude (delete) the KompasFlow project components from the KOMPAS-3D document, which is currently opened.

Solution








To control computation of the KompasFlow project.

	Start solve	<p>Start new calculation of KompasFlow project from scratch with removing the saved data.</p> <p>When starting a calculation from the beginning, all previously calculated data is deleted, so executing this command requires confirmation in a dialog box with the message "The previous calculation data will be lost! Continue?".</p>
	Open the solution	Load saved calculation data.
	Delete the solution	Delete saved calculation data.
	Continue	Continue previously stopped calculation.
	Stop	Stop the calculation. The calculation data will be saved and can be continued later.
	Finish	Finish the calculation and save the calculated data.

	Disable layers update	Stop updating visualization layers . This will save computational resources by not wasting them on rendering Layers . To activate updating the Layers , click this icon again.
---	-----------------------	--

Task statement






To move the input focus in the project tree on the appropriate element that is used to define the problem setting, and opens the property bar of this element.

	Substance	To create a new Substance (open <i>Substance creation</i> window)
	Adaptation	To create a new Adaptation of the computational grid.
	Time step	Go to Time Step in the project tree and open its properties window
	Processes	Go to Processes in the project tree and open its properties window
	Initial conditions	Go Initial conditions in the project tree and open its properties window
	Limiters	Go to Limiters in the project tree and open its properties window
	Computational grid	Go to Processes in the project tree and open its properties window

Boundary conditions





To create a [Boundary condition](#) with specified type.

If before creating the Boundary condition you select one or several faces of the geometry model (groups of facets) in the graphic area of the KOMPAS-3D's window, the new Boundary condition will be created on the appropriate Face list.

	Wall	To create a Wall type boundary condition.
	Symmetry	To create a Symmetry type boundary condition.
	Inlet/Outlet	To create a Inlet/Outlet type boundary condition.
	Non-reflecting	To create a Non-reflecting type boundary condition.
	Free outlet	To create a Free outlet type boundary condition.








Visualization

Commands in this group are used to create [visualization layers](#).

	Color contours	To create a Color contours type visualization layer.
	Vectors	To create a Vectors type visualization layer.
	Streamlines	To create a Streamlines type visualization layer.
	Computation grid section	To create a Computation grid section type visualization layer.






Results

Commands in this group are used to create [calculation results](#).

	Temperature	To create Temperature type calculation result (click) To select an option to create a calculation result (click and hold): T Temperature T' Total temperature
	Pressure	To create Pressure type calculation result (click) To select an option to create a calculation result (click and hold): P Pressure P' Total pressure
	Force	To create Force type calculation result (click) To select an option to create a calculation result (click and hold): F Force M_h Torque
	Velocity	To create Velocity type calculation result (click) To select an option to create a calculation result (click and hold): V Velocity M Mach number
	Heat flux	To create Heat flux type calculation result (click) To select an option to create a calculation result (click and hold): Q Heat flux J Radiation flux
	Mass flow	To create Heat flux type calculation result (click) To select an option to create a calculation result (click and hold): Q_M Mass flow Q_V Volume flow
	Density	To create Density type calculation result (click).

Additional commands

Using the commands in this group, you can open monitoring and optimization windows, check the geometry, access the help information and create reports.

	Open monitor	To open Monitoring window .
	Optimization params	To open Optimization parameters table in Parameters pane.
	Check geometry	To check the geometry. Activating this command during the calculation process will stop the calculation with loss of unsaved data.
	Update model geometry	To update the geometry of the model. This command is necessary if the geometry of a component has been changed in another Kompas module. Otherwise error message appears: "Model geometry may have changed; manual update required".
	Help KompasFlow	To open KompasFlow user guide.

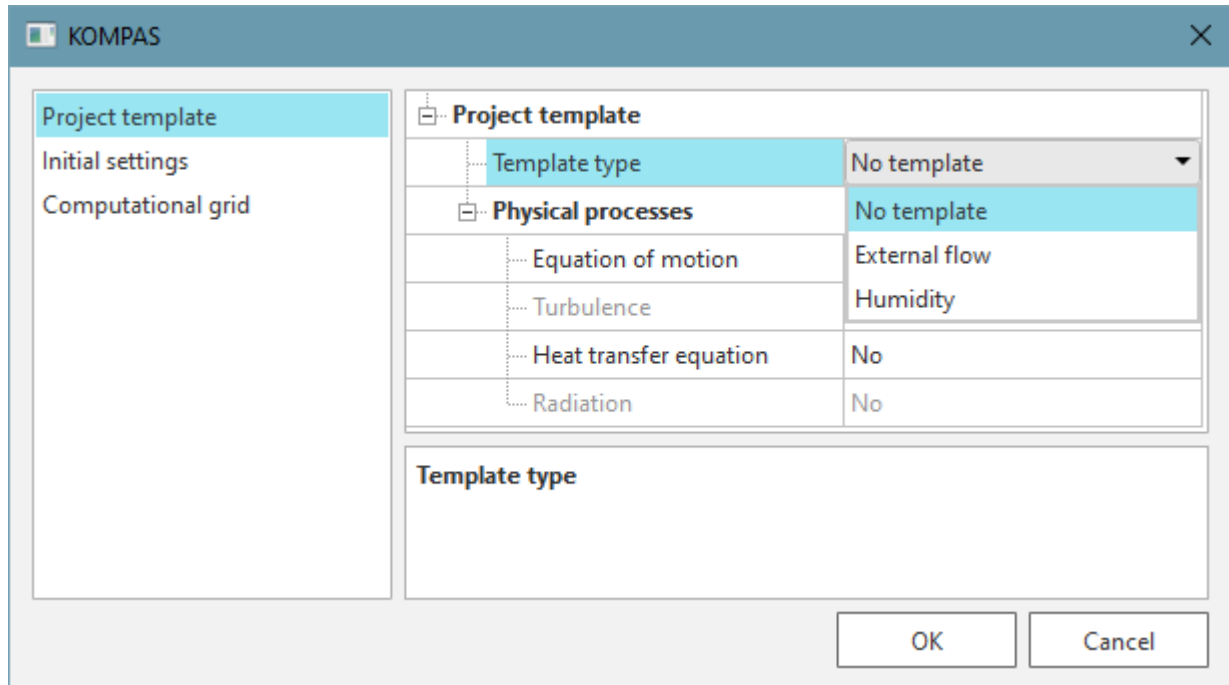


Create report

To create an [automatic report for the current project](#).

5.2 Wizard

When creating a new project, the Project creation wizard (Wizard) window appears.



The wizard window consists of 3 areas:

1. The tab area. In this area, you can switch between tabs. The contents of the tabs will be displayed in the properties area.

The following tabs are available:

Project template is a tab where you can select a project template depending on the task.

Initial settings is a tab corresponding to parameters of the Initial settings item in the project window.

Computational grid is a tab corresponding to parameters of Computational grid folder in the project window.

2. The parameters area. This area corresponds to the properties window, where you can set the parameters of items.

All settings, except the template, can be changed later.

3. Information line. A description of line selected in the properties area appears here.



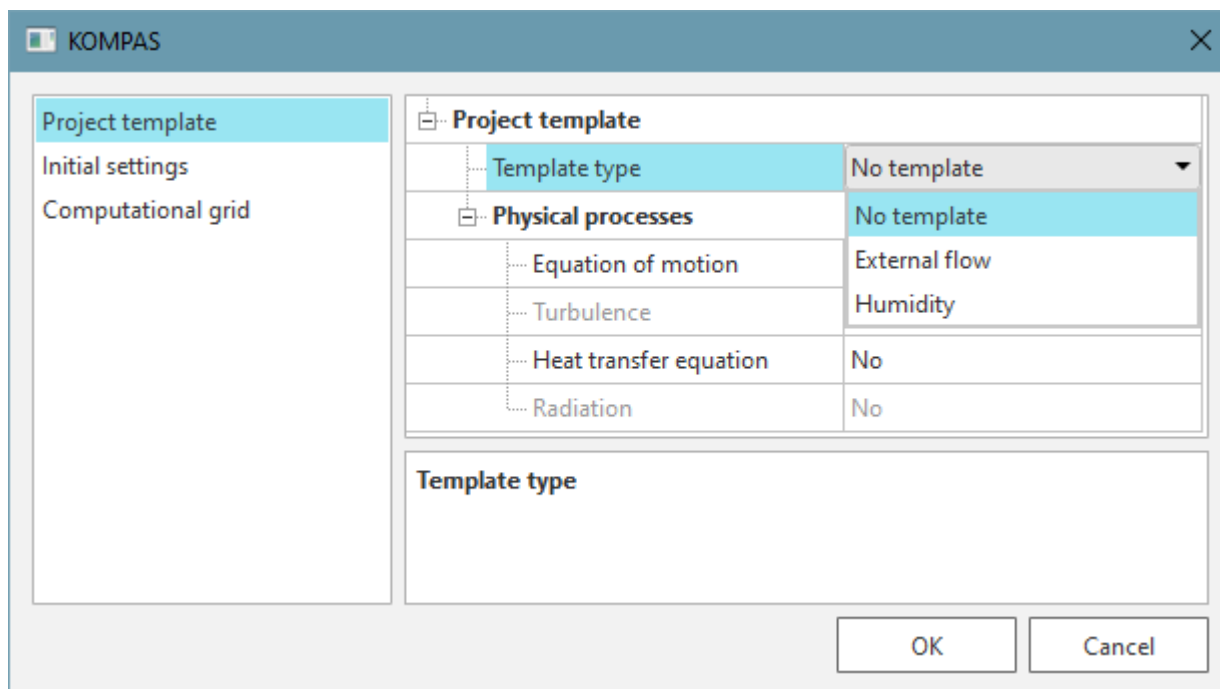
If you plan to create a project without calculating the external air or humidity, as well as adjust the project parameters later, then click OK to immediately proceed to the standard work on the project. For tasks with calculating external air flow or humidity, you must select the appropriate template in this window.

5.3 Templates



The template can be selected in the [project creation wizard](#) only when creating a new project!

It is not possible to change the template in created projects.



No template

A KompasFlow project with a standard set of settings.

When creating, you can specify the physical processes that will be used in the project. Activation of basic physical processes makes it possible to activate dependent physical processes (for example, to activate Turbulence, you must first activate the Equation of Motion). You can set the calculation grid settings.

External flow

A template for modeling the external blowing of an object.

In the template, the parameters of the air flow are set in **Initial settings**: speed, pressure, and temperature. You can also set the calculation grid settings.

In the project, a box is built around the object, which acts as a source of external ventilation. Its dimensions are set in **Comp.domain geometry**, in **Region boundaries** parameters group. Also, an **External BC** is automatically added to **Boundary conditions** folder, which type can be set as **Inlet/Outlet** or **Non-reflecting** for all surfaces of the box around the object.

Humidity

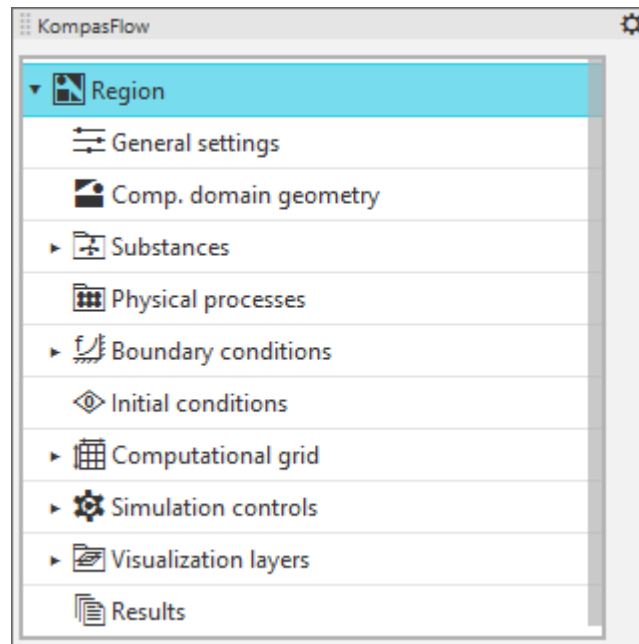
A template for calculating humidity or moisture content in the air.

This template uses strictly defined substances and physical processes that cannot be edited: **Air** and **Water Vapor** are used as substances, and all standard physical processes enabled, including **Mass transfer**. In the template, you can set initial conditions for speed, pressure and temperature, as well as humidity. You also can set parameters of calculation grid.

Two new types of boundary conditions become available in the project: **Source of moisture** and **Hood**, and 4 additional variables for **layers** and **results**: **Specific humidity**, **Absolute humidity**, **Relative humidity** and **Partial pressure**.

5.4 Items in the project window

Elements of the KompasFlow project tree are nodes of the hierarchical structure that is displayed in the KompasFlow tab of the KOMPAS-3D control panels:



Nodes containing other nodes are called folders or parent nodes. The nodes contained in the other nodes are called child nodes.

Each node may have a context menu that opens by mouse right clicking.

When a node is selected in the project tree, its parameters (properties) are displayed in the properties bar. Node properties are also arranged as a tree. Changing properties of a node is done by entering data into a field or selecting a value from a drop-down list.



Some properties of a node can be set only when it is created, in the **Parameters** tab of the KOMPAS-3D control panels.

The following sections contain details about user interface of elements in the project tree:

[Region](#)

[General settings](#)

[Computational domain geometry](#)

[Substance](#)

[Physical processes](#)

[Boundary conditions](#)

[Initial conditions](#)

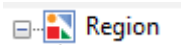
[Computational grid and adaptation](#)

[Computation parameters and limiters](#)

[Visualization layers](#)

Results (folder)

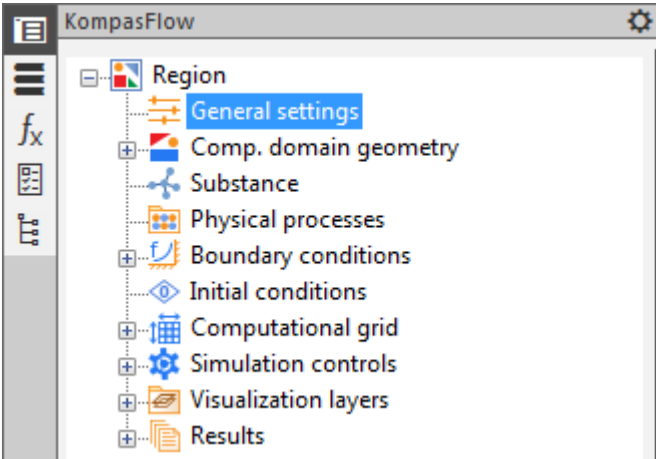
5.4.1 Region



The Region element is the global parent node of the KompasFlow project tree.



It has neither context menu nor parameters.

5.4.2 General settings



The General settings element contains the global settings of the KompasFlow project. It has no context menu.

Reference pressure	The reference pressure value P_{ref} , [Pa] and the reference temperature value T_{ref}
Reference temperature	[K]. In many cases, changes in pressure and temperature in the flow field due to hydrodynamic reasons are significantly less than the absolute values of pressure and temperature. Therefore, to increase the accuracy of the calculation, it is advisable to provide absolute values of pressure and temperature in the form of sums of reference and relative values: $P_{abs} = P_{ref} + P$ $T_{abs} = T_{ref} + T$ It is recommended to specify values of the reference pressure and temperature near the range, in which these physical variables vary. For simulations of processes that proceed at moderate pressures and temperatures it can be recommended to use the following reference values: $P_{ref} = 101000 \text{ [Pa]}$ $T_{ref} = 273 \text{ [K]}$ Selection of these values is useful because in this case the relative pressure corresponds to difference with the standard atmosphere pressure and the relative temperature practically corresponds to the Celsius temperature. And when an industrial power facility is simulated, you can specify reference values that correspond to the normal (design) pressure and temperature that are used in operation of the facility. In this case the relative values correspond to

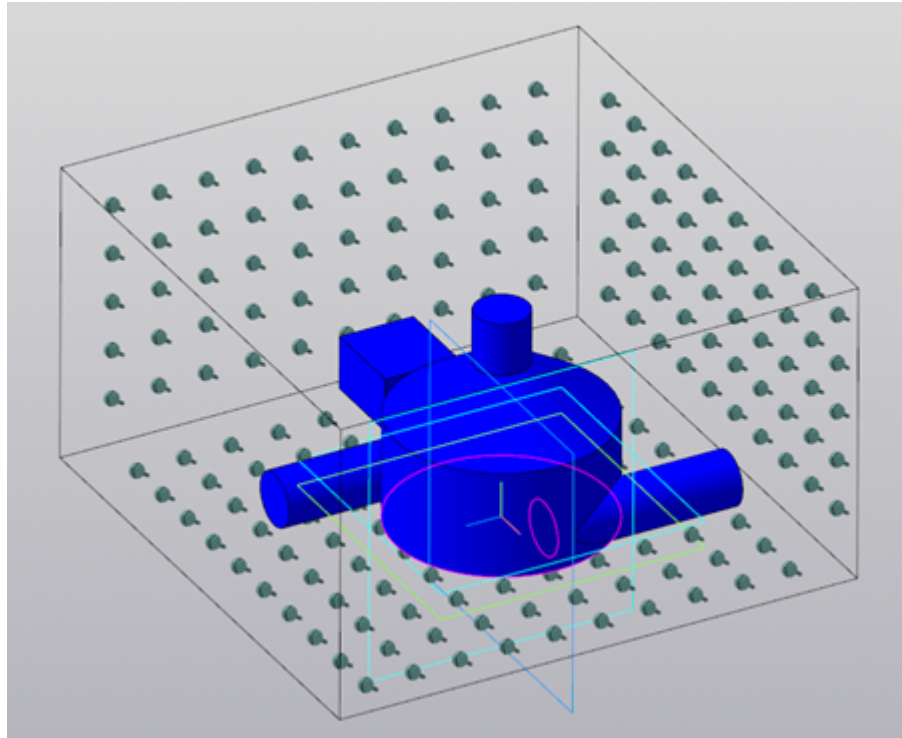
	deviations from the normal pressure and temperature.
Gravity vector	Components of the gravity vector along coordinate axes X, Y, Z, set in [ms-2]. The default values are zeros.
X	The gravity vector along the X-axis.
Y	The gravity vector along the Y-axis.
Z	The gravity vector along the Z-axis.
Hydrostatics	<p>Calculating taking into account hydrostatics. Counting hydrostatics raises the accuracy of calculation under conditions of natural convection or in the case of simulating liquids with a significant height of hydrostatic column.</p> <p>Available options are: Yes No.</p> <p>If Hydrostatics = Yes, then hydrostatics density parameters (g-Density settings) are available.</p>
	 Some node properties can only be set during creation, in the Settings tab of the KOMPAS-3D control panel area.
	 When visualizing the Pressure variable, static pressure values are shown without taking into account the hydrostatic component.
Tolerance of geom. calc.	<p>Tolerance of geometric calculations, [m], i.e., the accuracy of geometrical parameters (point positions of geometrical bodies in the project).</p> <p>The default value is 10^{-8}.</p>
External task	<p>External flow simulation.</p> <p>Yes — simulation of external flow is enabled in a project. External region boundary conditions becomes available.</p> <p>No — simulation of external flow is disabled.</p>
BC external region	<p>Boundary conditions, which are set on boundaries of the external region.</p> <p>This parameter is only available when simulating an external flow (External task = Yes).</p> <p>Nonreflect — Non-reflective type boundary conditions are set on external boundaries.</p> <p>Inlet/Outlet — Inlet/Outlet type boundary conditions are set on external boundaries.</p>

5.4.2.1 Simulating an external flow

When an external flow (the flow-around) is simulated, when **External task = Yes** is set in properties of the [General settings](#), an external region is to be specified.

The external region is an circumscribed parallelepiped (3D box) with user-defined sizes. The program will display, in its graphical window, boundaries of the external region and velocity vectors on boundaries of the region specified in the [Initial conditions](#).

Boundaries of the external flow along axes X, Y, Z are set by dimensionless values **Region boundaries > ...** in properties of the folder [Comp. domain geometry](#).

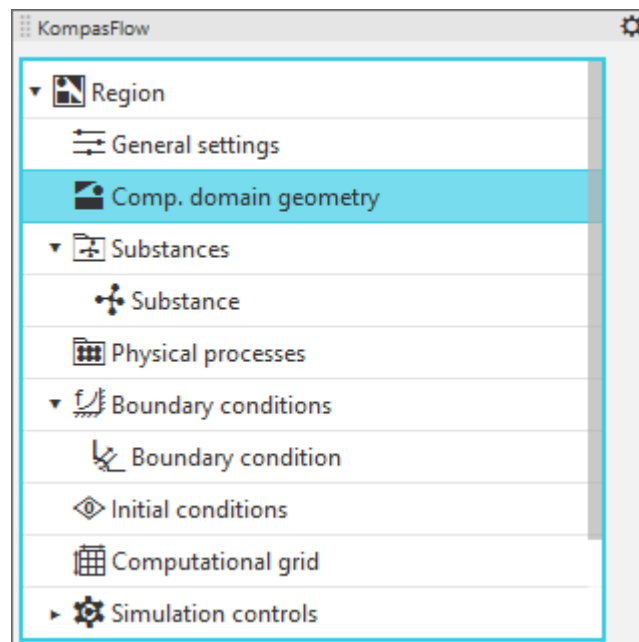


Simulating an external flow is available when you work with assemblies and you can use all functionality of the program as if you work with a detail.



When an external flow is simulated, equation of motion cannot be disabled (**Equation of motion = Yes** is always set in properties of **Physical processes**).

5.4.3 Computational domain geometry



Comp. domain geometry folder contains the **Bodies** subfolder that contains elements that correspond to geometrical bodies created by KOMPAS-3D and used by the KompasFlow application.

Context menu of the "Comp. domain geometry" folder

Selection of bodies	<p>Selecting bodies for the geometrical model. This command allows you to select a geometric body for the Region (computational domain) and built-in geometrical bodies.</p> <p>By default, the largest Body is selected as the Region.</p>
---------------------	--

! Names of **Bodies** in the project are to be different. When you form **Bodies** from standard parts, rename the **Bodies** to satisfy this condition

The **Comp. domain geometry > Bodies** folder has no context menu.

Comp. domain geometry > Bodies > Body elements have no context menus.

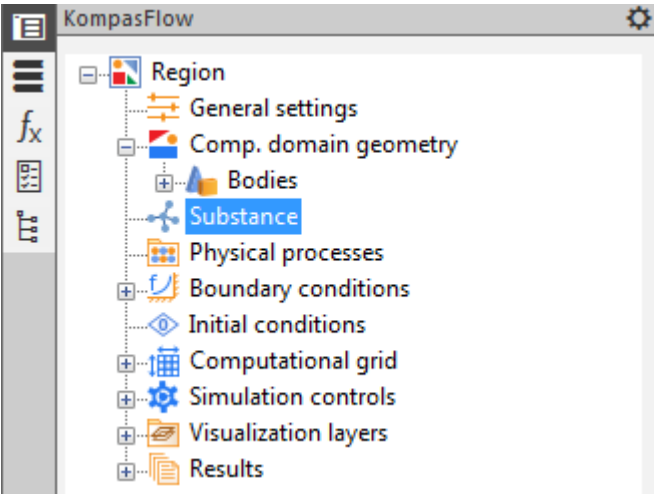
Parameters of the "Comp. domain geometry" folder

Region (Body)	A region (body) that is selected as the computational domain.
Convert dimensions	<p>Need to convert dimensions to meters.</p> <p>Available options: Yes No.</p>
Triangulation parameters	Group of parameters that specify the triangulation of the computational domain.
Angle	Limitation the angular deviation of surface, [degrees]
Sag	Limitation the linear deviation of the surface from the triangulation plate, [mm]
Edge length	Limitation the edge sizes for triangulation plate, [mm]
Region boundaries	Numerical values of boundaries of the external region that are specified when an external flow (the flow-around) is simulated.
X+	Dimensionless values defining sizes of the external region relating to sizes of the 3D geometry model of the object, around which the external flow is simulated.
X-	
Y+	
Y-	
Z+	
Z-	

Parameters of the "Comp. domain geometry > Bodies > Body" element

Boundary condition	Boundary condition , which is assigned to the whole surface of the selected Body. It is selected from a drop-down list that contains all Boundary conditions specified in the project.
--------------------	--


5.4.4 Substance

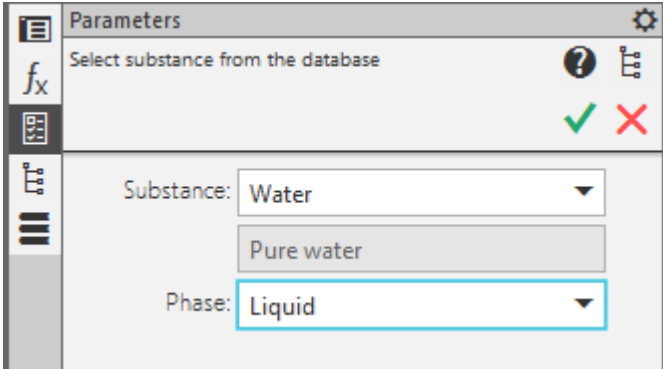


The Substance element sets the physical properties of the simulated medium. You can specify properties of the Substance in the following ways:

- load a Substance from the built-in substance database (using the Load from database command, see below)
- set physical properties of a Substance manually (just after creating the project, or using the Set manually command if some Substance has been loaded from the database, see below)
- first specify some individual physical properties of Substance manually, then load any Substance from the base and select User-specific: (value) from a drop-down list for some of the parameters in the properties bar.

Context menu

Load from database Load Substance from the database where properties of many substances are stored. After applying this command, the Parameters tab will open, where you will need to select Substance and its state of aggregation (in the Phase field), and then click the  icon:



See also: [Substance and its parameters](#).



Some Substances loaded from the base of substances may have several options for one state of aggregation. For example, gas Air can be set in two options: either dissociating or non-dissociating.

Set manually This command allows you to specify physical properties of the Substance manually:


- just after creating the project.
- or, if some Substance has already been loaded from the database of substances.



Applying the **Set manually** command does not change the database of substances.




At next loads of a **Substance** from the database, it is possible to use previously entered custom values of physical parameters instead of values from the database.

You can also create a Substance by clicking the icon  in the Task statement command set in the KompasFlow toolbar.



Substance	
Name	Water - Liquid
State of aggregation	Liquid
Density	From database
Molar mass	970 ▼ ...
Viscosity	From database
Thermal conductivity	From database
Specific heat	From database

Parameters

Name	This parameter is available only for Substances loaded from the database. Name of the Substance is formed automatically and contains the state of aggregation (phase).
State of aggregation	State of aggregation of the substance. Possible values: Gas Liquid Solid.
Ideal gas law	Gas density can be set either by a constant or by ideal gas law. Possible values: <ul style="list-style-type: none"> • Yes - ideal gas law is applied. • No - gas density is set by a constant This parameter is available when State of aggregation = Gas (see above).
Reference temperature	Reference temperature is displayed when Ideal gas law = Yes and Heat transfer equation = No in the properties of the Physical processes element.
	 Reference temperature is displayed here for information only; it is set in properties of the General settings element.
Density*	Density of the Substance, [kg/m ³]. When Ideal gas law = Yes, the Density parameter is absent (see above).
Molar mass*	Molar mass of the Substance, [kg/mole]
Viscosity*	Viscosity of the Substances, [Pa s] = [kg·m ⁻¹ ·s ⁻¹]. The Viscosity parameter is absent when State of aggregation = Solid (see above).
Thermal conductivity*	Thermal conductivity of the Substance [W/(m·K)] = [kg·m·s ⁻³ ·K ⁻¹]
Specific heat*	Specific heat capacity of the Substance [J kg ⁻¹ ·K ⁻¹] = [m ² ·s ⁻² K ⁻¹]

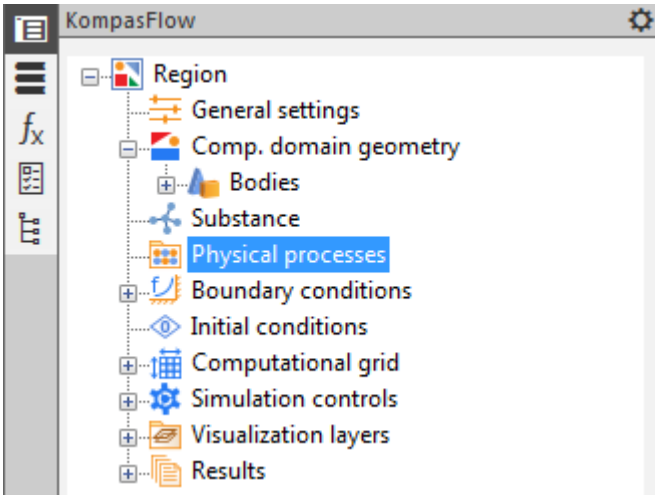
*If Substance was loaded from the substance database, you can select in this field:

- **From database** - in this case, the value from the database will be used
- **User value:** (value) - the manually set value will be used

If the Substance has been loaded from the substance database, then, in the selected numerical field is in the properties bar, the  screen button appears. Clicking the  button opens, in the parameters pane, the table with values received from the substance database. Example:


#	P	T	F(P,T)
1	10132.5	100	7.3413...
2	10132.5	110	8.0363...
3	10132.5	120	8.7109...
4	10132.5	130	9.3655...
5	10132.5	140	1e-5
6	10132.5	150	1.06e-5
7	10132.5	160	1.12e-5
8	10132.5	170	1.18e-5

5.4.5 Physical processes



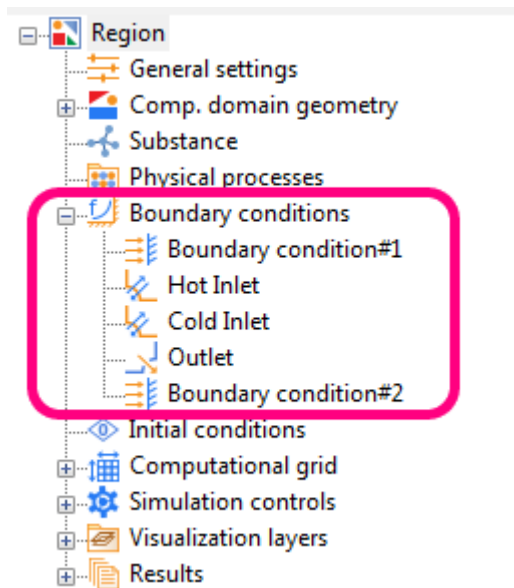
The Physical processes element specifies the set of the solvable equations and their settings.

The Physical processes element has no context menu.

Parameters	
Equation of motion	<div><div>This parameter defines whether or not the equation of motion (Navier-Stokes equation) will be solved.</div><div>Available options: Yes No.</div></div>
	<div><div></div><div>When an external flow (the flow-around) is simulated you cannot disable equation of motion (Equation of motion = Yes will be always set in this case).</div></div>
Turbulence	<div><div>The turbulence model being used. Available options:</div><div><ul style="list-style-type: none">• No - the program will not use a turbulence model• Yes - the program will use the standard k-ε turbulence model</div><div>Simulating of turbulence is performed with using wall functions.</div></div>

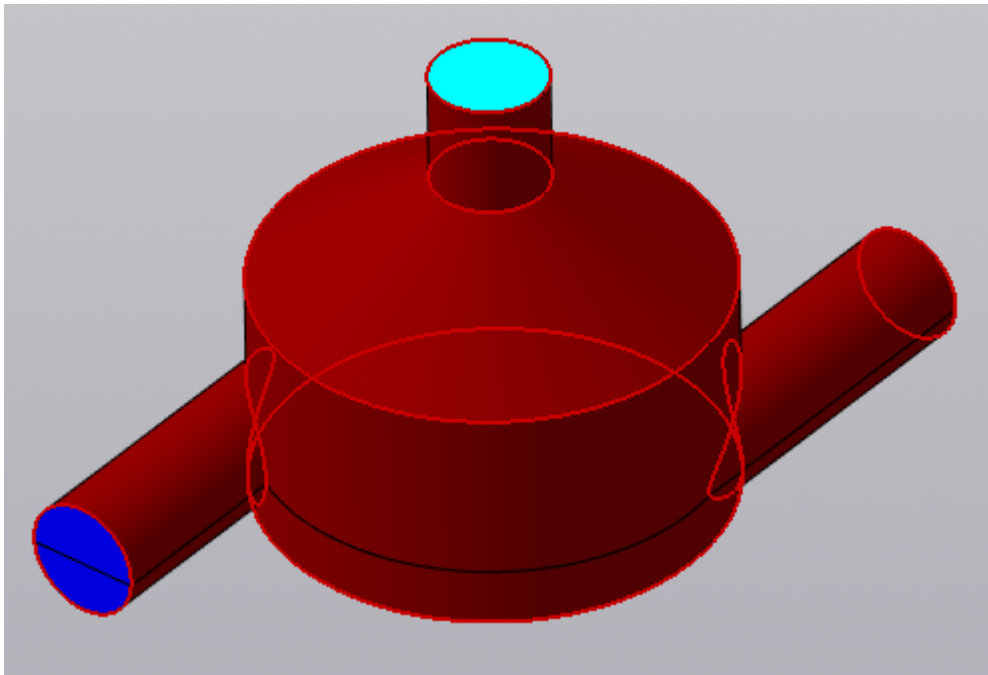
	<p>When you specify Inlet/Outlet boundary conditions and Initial conditions, you set the Turbulence level (available options are: Low, 3% Medium, 10% High, 20%).</p> <p>The turbulence scale is set by the program automatically as 1% of the minimal size of the computational domain (without taking into account the direction, along which the computational domain contains only one cell, if such direction exists).</p>
Heat transfer equation	<p>This parameter defines whether or not the equation of heat transfer (except radiation flux) will be solved.</p> <p>Simulating of the radiation flux is set by the Radiation parameter (see below).</p> <p>Available options: Yes No.</p>
Radiation	<p>Simulating of the radiation heat flux. Available options: Yes No.</p> <p>The program applies the following parameters of optically transparent medium:</p> <ul style="list-style-type: none"> • Refraction index = 1 • Absorption coefficient = 10 • Relaxation coefficient = 0.2

5.4.6 Boundary conditions








The **Boundary conditions** folder contains elements **Boundary condition #N** that correspond to the boundary conditions of the simulated flow or heat transfer.







The Boundary condition, which is selected in the project tree, is highlighted in the graphical area with maroon color and/or red contours (depending on settings of model displaying):



Types of boundary conditions

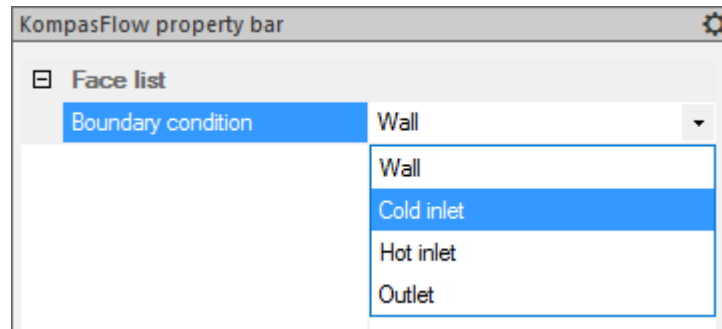
	Symmetry	Symmetry type boundary condition.
	Wall	Wall type boundary condition.
	Inlet/Outlet	Inlet/Outlet type boundary condition.
	Free outlet	Free outlet type boundary condition.
	Non-reflecting	Non-reflecting type boundary condition.

Context menu

Create BC	This command creates a new Boundary condition #N. After applying this command, a new child element Boundary condition #N with default properties will be created.
<div> You can also create a Boundary condition by clicking an appropriate icon (, , , , ) in the Boundary conditions command set in the KompasFlow toolbar.</div>	

Assigning boundary conditions to faces of the geometry model

When you select one or several faces of the geometry model (groups of facets) in the graphic area of the KOMPAS-3D's window, the KompasFlow property bar contains the Face list > Boundary condition parameter that specifies the boundary condition, which is assigned to the face(s):



Assigning boundary conditions to boundaries of the external region

When [an external flow \(the flow-around\) is simulated](#), the following boundary conditions can be set on boundaries of the external region:

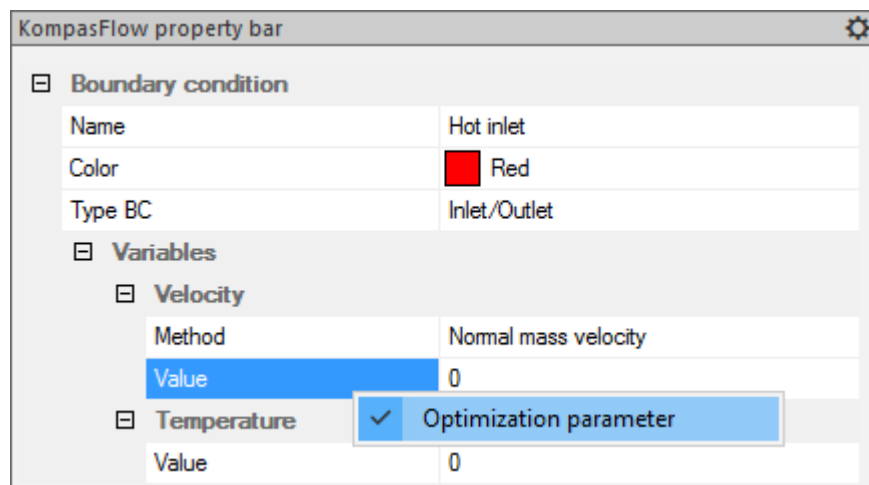
- [Non-reflecting](#)
- [Inlet/Outlet](#)

This is specified by the BC external region parameter in properties of the [General settings](#) element.

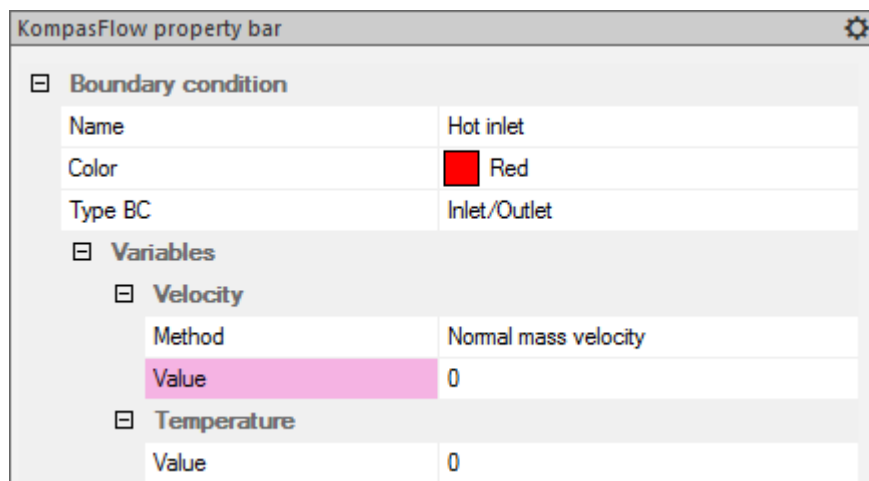
Declaring a numerical property of some Boundary conditions as an optimization parameter

Numerical parameters of Boundary conditions can be declared as [optimization parameters](#).

To do so, you have to open the context menu of a parameter and enable the Optimization parameter option:



Optimization parameters are highlighted in the properties of Boundary conditions with pink color:



5.4.6.1 Symmetry

Context menu commands

Remove BC	Delete the selected Boundary condition #N.
Copy	Copy the selected Boundary condition #N.
Create Result	Create Result for the selected Boundary condition #N.

Parameters

Name	<p>The name of the Boundary Condition that is displayed in the project tree.</p> <p>By default, the name is assigned automatically when creating a new BC: A boundary condition with an ordinal number.</p> <p>The name can be changed in the future. For convenience, it is recommended to enter a name corresponding to the purpose or location of the control unit, for example: Wing, Air Intake, Window, etc.).</p>
Color	<p>The color for displaying the Boundary condition in the graphical area.</p> <p>You can select a color from the drop-down list of standard colors or select a custom color using the Color = Other...</p>
Transparency	Transparency adjustment by a fractional value in the range from 0 to 1.
Type BC	<p>Type of boundary condition.</p> <p>When you change the type of boundary condition, available parameters in properties window change.</p> <p>Only the parameters that can be set for the selected type of BC will be available.</p>

Formulas

The specified velocity on a Symmetry boundary condition is set automatically:

Velocity formulas:

$$V_{b,n} = 0$$

$$V_{b,\tau} = |\mathbf{V}_{c,abs} - \mathbf{V}_B|$$

Here:

$\mathbf{V}_{c,abs}$ is the absolute velocity of liquid in the center of a boundary cell.

Temperature formulas:

$$\left. \frac{\partial T}{\partial y} \right|_b = 0$$


$$T_b = T_{cell}$$




5.4.6.2 Wall

Context menu commands

Remove BC	Delete the selected Boundary condition #N.
Copy	Copy the selected Boundary condition #N.
Create Result	Create Result for the selected Boundary condition #N.

Parameters

Name	<p>The name of the Boundary Condition that is displayed in the project tree.</p> <p>By default, the name is assigned automatically when creating a new BC: A boundary condition with an ordinal number.</p> <p>The name can be changed in the future. For convenience, it is recommended to enter a name corresponding to the purpose or location of the control unit, for example: Wing, Air Intake, Window, etc.).</p>
Color	<p>The color for displaying the Boundary condition in the graphical area.</p> <p>You can select a color from the drop-down list of standard colors or select a custom color using the Color = Other...</p>
Transparency	Transparency adjustment by a fractional value in the range from 0 to 1.
Type BC	<p>Type of boundary condition.</p> <p>When you change the type of boundary condition, available parameters in properties window change.</p> <p>Only the parameters that can be set for the selected type of BC will be available.</p>
Roughness	The equivalent sand roughness of the wall, [μm]
Variables	Variables used in this boundary condition.
Temperature	Group of parameters for Temperature variable.
Method	<p>Method of specifying the boundary condition for the temperature.</p> <p>Available options: Constant Heat flux Radiation equilibrium External heat exchange</p>
	<p> When the physical process Radiation is enabled, the following methods for Temperature variable cannot be set:</p> <ul style="list-style-type: none"> • Radiation equilibrium • External heat exchange
Value	Here you specify the value of Temperature, [K], or value of the specific Heat flux, [W/m^2].
Energy flux	<p>These parameters are set when Method = Radiation equilibrium is selected.</p> <ul style="list-style-type: none"> • Energy flux is the total energy flux from the surface of the boundary condition into the gas or liquid, $J_{q,b}$ [W/m^2].

Blackness	<ul style="list-style-type: none"> Blackness is the emissivity factor of the surface of the boundary condition (emissivity ε_w); it is specified by a non-dimensional value in the range [0, 1].
T_Inf	<ul style="list-style-type: none"> T_Inf is the relative temperature at infinity $T_{\infty,abs}$, [K], that defines the radiative flux falling on the surface of the boundary condition. <p>It is assumed that diffusion and radiative energy fluxes are directed normally to the surface. Surface temperature could be found by the iterative solution of the</p> $J_{q,b} = \left(\lambda + C_p \frac{\mu_t}{Pr_t} \right) \frac{T_{w,abs} - T_{c,abs}}{y_c} +$ <p>equation: $+ \sigma_{rad} \varepsilon_w (T_{w,abs}^4 - T_{\infty,abs}^4)$</p> <p>where y_c is distance from the surface to the center of wall cell. The user sets values $J_{q,b}$, ε_w and $T_{\infty,abs}$.</p> <p>When the program calculates a Result for the Heat flux, the radiative component $\sigma_{rad} \varepsilon_w (T_{w,abs}^4 - T_{\infty,abs}^4)$ is not taken into account.</p>
T of external medium	These parameters are set when Method = External heat exchange is selected. Here you specify:
Blackness	<ul style="list-style-type: none"> relative temperature of the external medium (T of external medium), [K]
Number of layers	<ul style="list-style-type: none"> the external heat transfer coefficient (Heat-transfer coef.), [W m⁻² K⁻¹]
Layer thickness	<ul style="list-style-type: none"> the emissivity factor ε_w of the surface of the boundary condition (Blackness) number of heat transferring layers (Number of layers)
Thermal conductivity of the layer	<ul style="list-style-type: none"> thickness of the N-th heat transferring layer (Layer N > Layer thickness), [m] thermal conductivity of the Nth heat transferring layer (Layer N > Thermal conductivity of the layer), [W/(m·K)]
Radiation	
Method	<p>Method of specifying the radiation energy flux from the surface of the boundary condition.</p> <p>Available options:</p> <ul style="list-style-type: none"> Density heat flux – in the parameter Variables > Radiation > Energy flux (see below) you specify density of the radiation heat flux. When a boundary condition is created, this option is marked by the icon . Heat flux – in the parameter Variables > Radiation > Energy flux (see below) you specify the absolute (total) value of the radiation flux from the surface of the boundary condition. When a boundary condition is created, this option is marked by the icon . Auto – automatic calculation of the radiation flux based on the user-defined Blackness of the surface of the boundary condition (see below). When a boundary condition is created, this option is marked by the icon .
Energy flux	<p>The user-defined value of either Density heat flux (specific radiation flow), [W/m²] or Heat flux (absolute radiation flux), [W].</p> <p>This parameter is only available when Variables > Radiation > Method = Density heat flux Heat flux.</p>
Blackness	<p>Blackness of the surface of the boundary condition. It is used for automatic calculation of the radiation flux.</p> <p>This parameter is only available when Variables > Radiation > Method = Auto.</p>

Formulas

For Velocity at the wall, this condition is set automatically:

$$V_b = 0$$

For Temperature, the condition depends on the value, which is set for the parameter Variables > Temperature > Method:

If Method = Constant, then the value of Temperature is set by the user:

$$T_b = T_{user}$$

If Method = Heat flux, then the user specifies the value set of the heat flux:

$$J_{q,b} = J_{user}$$



When the turbulence is simulated on the **Wall** boundary condition, the wall functions are used (an empirical ratio that makes possible considering the nonlinear nature of the velocity profile in the near-wall layer).

5.4.6.3 Inlet/Outlet

Context menu commands

Remove BC	Delete the selected Boundary condition #N.
Copy	Copy the selected Boundary condition #N.
Create Result	Create Result for the selected Boundary condition #N.

Parameters

Name	<p>The name of the Boundary Condition that is displayed in the project tree.</p> <p>By default, the name is assigned automatically when creating a new BC: A boundary condition with an ordinal number.</p> <p>The name can be changed in the future. For convenience, it is recommended to enter a name corresponding to the purpose or location of the control unit, for example: Wing, Air Intake, Window, etc.).</p>
Color	<p>The color for displaying the Boundary condition in the graphical area.</p> <p>You can select a color from the drop-down list of standard colors or select a custom color using the Color = Other...</p>
Transparency	Transparency adjustment by a fractional value in the range from 0 to 1.
Type BC	<p>Type of boundary condition.</p> <p>When you change the type of boundary condition, available parameters in properties window change.</p> <p>Only the parameters that can be set for the selected type of BC will be available.</p>
Variables	Variables used in this boundary condition.
Velocity	Parameters for Velocity variable.
Method	Method of specifying the boundary condition for the Velocity variable.

	Available options: Normal mass velocity Normal velocity with pressure Velocity with pressure Inlet pressure Total pressure Fixed velocity Mass flow. See details in the subsection "Methods for specifying the velocity" below.
Value	Normal specific mass flow rate, [kg/(m ² ·s)]. It is set when Variables > Velocity > Method = Normal mass velocity.
Velocity	Normal flow rate [m/s]. It is set when Variables > Velocity > Method = Velocity with pressure.
X	Components of the flow rate vector along axes X, Y and Z, [m/s].
Y	It is set when Variables > Velocity > Method = Velocity with pressure Fixed velocity.
Z	
Pressure	Depending on the selected Method of specifying Velocity, this is static pressure, or relative pressure at inlet, or total pressure, [Pa].
Total pressure	It is set when Variables > Velocity > Method = Normal velocity with pressure Velocity with pressure Inlet pressure or Variables > Velocity > Method = Total pressure.=.
Mass flow	Mass flux through the boundary condition, [kg/s]. Set by Variables > Velocity > Method = Mass flow.
Temperature	Temperature of the flow, [K]. This parameter is available only when Heat transfer equation = Yes is set in Physical processes .
Turbulence level	This is an indicative level of turbulence. Available options: Low, 3% Medium, 10% High, 20%. This parameter is available only when Equation of motion = Yes and Turbulence = Yes is set Physical processes .

Methods for specifying the velocity

Below are descriptions of how the program understands various values of the parameter Variables > Velocity > Method.

1. Normal mass flow rate

When this method is selected, the Variables > Velocity > Value parameter is used to specify the normal mass flow rate, [kg/(m²·s)]:

$$\rho V_n|_b = \rho V_n|_{user}$$

2. Normal velocity with pressure

When this method is selected, the normal mass flow rate [m/s] is set, and the Variables > Velocity > Pressure parameter is used to specify the static pressure, [Pa]:

$$V_{n,b} = V_{user} > 0$$

$$P_b = P_{user}$$

This condition assumes an inlet. The user sets the velocity's absolute value and static pressure in an undisturbed flow (at infinity).

The absolute velocity is calculated in the program as sum of the relative velocity of the liquid V_b and the local velocity of the boundary V_B . The negative value of the velocity (forced suction) cannot be set. However, the Normal velocity with pressure condition allows outflow through the inlet. In such a case, the boundary condition will work the same as a Total pressure condition.

The following situations are implemented:

Inlet, supersonic:

Values of normal velocity and pressure on the boundary are fixed and are equal to values at infinity. Velocity is directed along the inward-directed normal to the inlet surface. The Temperature parameter specifies the static temperature.

Inlet, subsonic:

The velocity's absolute value on the boundary is equal to the velocity's absolute value in the center of the border cell. The velocity vector is directed into the computational domain normally to the boundary. Velocity in the center of the cell is calculated from the solution. Static pressure is calculated from a condition of constancy of the total pressure determined by the specified values of static pressure, velocity and static temperature according to the following formulae:

$$P_{tot} = P + \rho \frac{V_{b,abs}^2}{2} \quad \text{if the Substance has State of aggregation = Liquid}$$

$$P_{tot} = P \left(\frac{T_{tot}}{T} \right)^{\frac{\gamma}{\gamma-1}} \quad \text{if the Substance has State of aggregation = Gas}$$

Outlet, supersonic:

The velocity vector on the boundary is equal to the velocity vector in the center of the border cell. Static pressure on the boundary is equal to the static pressure in the center of the border cell.

Outlet, subsonic:

The velocity vector on the boundary is equal to the velocity vector in the center of the border cell.

Static pressure on the boundary is equal to the total pressure determined by values Puser, Vuser, Tuser.

Velocity with pressure

When this method is selected, parameters Variables > Velocity > Velocity > X, Y, Z are used to specify components of the flow rate vector along axes X, Y and Z, [m/s], and the parameter Variables > Velocity > Pressure is used to specify static pressure, [Pa]. Negative value of the velocity's projection on the inward-directed normal to the inlet surface (forced suction) cannot be set.

This condition is used for specifying a fluid's inflow. If during the simulation a whirl approaches the BC and conditions of the fluid's outlet through the BC appear, then anyway this BC will work correctly (see descriptions of situations "Outlet, supersonic" and "Outlet, subsonic" below).

The user sets the velocity vector:

$$\mathbf{V}_b = \mathbf{V}_{user}, \quad V_n|_b = \mathbf{V}_b \cdot \mathbf{n} > 0$$

and static pressure in the undisturbed flow (at infinity):

$$P_b = P_{user}$$

The absolute velocity is calculated in the program as the sum of the relative velocity of liquid \mathbf{V}_b and the local velocity of the boundary \mathbf{V}_B . The negative value of the velocity projection on an inward-directed normal to the inlet surface (forced suction) cannot be set. However, the Velocity with pressure condition allows outflow exit through the inlet. In this case, it will operate the same as the Total pressure condition.

The following situations are implemented:

Inlet, supersonic:

Values of velocity components and pressure on the boundary are fixed and equal to values at infinity. In the Temperature parameter, a static temperature is set.

Inlet, subsonic:

The velocity's absolute value on the boundary is equal to the velocity's absolute value in the center of a border cell. The velocity vector is collinear to the velocity vector that is set in the interface. Velocity in the center of the cell is calculated from the solution. The static pressure is calculated from a condition of constancy of the total pressure, determined by the specified values of static pressure, velocity and static temperature, according to the following formulae:

$$P_{tot} = P + \frac{\rho V_{b,abs}^2}{2}, \text{ if the Substance has State of aggregation = Liquid}$$

$$P_{tot} = P \left(\frac{T_{tot}}{T} \right)^{\frac{\gamma}{\gamma-1}}, \text{ if the Substance has State of aggregation = Gas}$$

Outlet, supersonic:

The velocity vector on the boundary is equal to the velocity vector in the center of a border cell. Static pressure on the boundary is equal to the static pressure in the center of a border cell.

Outlet, subsonic:

The velocity vector on the boundary is equal to the velocity vector in the center of a border cell. Static pressure on the boundary is equal to the total pressure determined by values P_{user} , V_{user} , T_{user} .

Inlet pressure

When this method is selected, the relative pressure at the inlet is specified, [Pa]:

$$P_b = P_{user}$$

The velocity vector at the given boundary equals to that in the center of the corresponding near-boundary cell. The cell-center velocity results from integration of the momentum equation. The given boundary condition (BC) can be specified at subsonic inlet and subsonic outlet.

The following situations are implemented in the program:

- Outlet, supersonic flow: The static pressure at the boundary equals to that in the cell center. The pressure value, specified in the interface, is ignored.
- Outlet, supersonic flow: The static pressure at the boundary equals to the value specified in the interface.
- Inlet, subsonic flow: The static pressure at the boundary equals to the value specified in the interface.

If only one inlet and one outlet are presented in the simulated problem, then specifying BC Inlet pressure both at the inlet and at the outlet is incorrect from mathematical point of view. In this case it will be also incorrect to specify BC Inlet pressure at the inlet and pressure at the outlet.

An example of correct combination is Inlet pressure (at inlet) + Normal mass velocity (at outlet). In this problem setting the normal mass velocity at outlet is to be negative (outflow).

Total pressure

When this method is selected, the total pressure, [Pa], is specified. The Total pressure can be set on a subsonic inlet, a supersonic outlet and a subsonic outlet.

The user sets the total pressure $P_{tot,b} = P_{user}$.

Fixed velocity

When this method is selected, components of the flow rate vector along axes X, Y and Z, [m/s] are specified. This condition can be set on a subsonic inlet and on a subsonic outlet. The user sets a velocity vector, which doesn't change in during the calculation:

$$\mathbf{V}_b = \mathbf{V}_{user}$$

if $\mathbf{V}_b \cdot \mathbf{n} > 0$, then inlet is implemented.

if $\mathbf{V}_b \cdot \mathbf{n} < 0$, then outlet is implemented (the surface normal is directed into the computational domain).

It is recommended to use Fixed velocity for solving problems, in which distribution of the velocity is known (usually on inlet). It should be noted that, when a compressible flow is simulated (when density depends on pressure) using this boundary condition, the flow through the surface, on which the BC is set, will not be fixed. It will depend on the static pressure that is found near this surface during the solving. In the case of a mobile boundary, the velocity of the fluid, set by this method, is relative. Absolute velocity is calculated in the program

as the sum of the relative velocity of the fluid \mathbf{V}_b and the local velocity of the boundary \mathbf{V}_B .

Mass flow

When this method is selected, mass flow of the substance through a surface of the boundary condition, [kg/s], is set.

Specifics of velocity representation methods "Normal velocity with pressure" and "Velocity with pressure"



Method = Normal velocity with pressure and **Method = Velocity with pressure** in projects with subsonic flow do not provide on the boundary those values of velocity and pressure that have been set. Velocity and pressure for these boundary conditions in the subsonic mode are recalculated through the total pressure. It is recommended to specify **Method = Normal velocity with pressure** and **Method = Velocity with pressure** for supersonic flows (then velocity and pressure will be the same as those that were set on the boundary condition).

5.4.6.4 Free outlet

The velocity vector on the boundary is equal to the velocity vector in the center of a border cell. Velocity in the center of the cell is calculated from the solution. The Free outlet boundary condition cannot be set on a subsonic inlet. The following situations are implemented:

- Outlet, supersonic: Static pressure on the boundary is equal to the static pressure in the center of the border cell. Pressure that is set in the interface is ignored.
- Outlet, subsonic: Static pressure on the boundary is equal to the static pressure that is set in the interface.
- Inlet, subsonic: Static pressure on the boundary is equal to the static pressure that is set in the interface.



Free outlet boundary condition cannot be set at the subsonic inlet.

Context menu commands

Remove BC	Delete the selected Boundary condition #N.
Copy	Copy the selected Boundary condition #N.
Create Result	Create Result for the selected Boundary condition #N.

Parameters

Name	<p>The name of the Boundary Condition that is displayed in the project tree.</p> <p>By default, the name is assigned automatically when creating a new BC: A boundary condition with an ordinal number.</p> <p>The name can be changed in the future. For convenience, it is recommended to enter a name corresponding to the purpose or location of the control unit, for example: Wing, Air Intake, Window, etc.).</p>
Color	<p>The color for displaying the Boundary condition in the graphical area.</p> <p>You can select a color from the drop-down list of standard colors or select a custom color using the Color = Other...</p>
Transparency	Transparency adjustment by a fractional value in the range from 0 to 1.
Type BC	<p>Type of boundary condition.</p> <p>When you change the type of boundary condition, available parameters in properties window change.</p> <p>Only the parameters that can be set for the selected type of BC will be available.</p>
Variables	Variables used in this boundary condition.
Velocity	Parameters for Velocity variable.
Pressure	<p>User-defined static pressure value, [Pa]:</p> $P_b = P_{user}$

Temperature	Parameters for Temperature variable.
Value	<p>Temperature, [K], for a situation when there is an inverse inflow through the Free outlet.</p> <p>In the event of outflow from the computational domain, the temperature zero gradient condition is applied:</p> $\left. \frac{\partial T}{\partial y} \right _b = 0$ $T_b = T_{cell}$

5.4.6.5 Non-reflecting

Non-reflecting boundary condition can be set on inlet and on outlet. You have to specify components of velocity, static pressure, and temperature at infinity.



The Non-reflecting boundary condition can be used only for Mach numbers larger than 0.1.

Context menu commands

Remove BC	Delete the selected Boundary condition #N.
Copy	Copy the selected Boundary condition #N.
Create Result	Create Result for the selected Boundary condition #N.

Parameters

Name	<p>The name of the Boundary Condition that is displayed in the project tree.</p> <p>By default, the name is assigned automatically when creating a new BC: A boundary condition with an ordinal number.</p> <p>The name can be changed in the future. For convenience, it is recommended to enter a name corresponding to the purpose or location of the control unit, for example: Wing, Air Intake, Window, etc.).</p>
Color	<p>The color for displaying the Boundary condition in the graphical area.</p> <p>You can select a color from the drop-down list of standard colors or select a custom color using the Color = Other...</p>
Transparency	Transparency adjustment by a fractional value in the range from 0 to 1.
Type BC	<p>Type of boundary condition.</p> <p>When you change the type of boundary condition, available parameters in properties window change.</p> <p>Only the parameters that can be set for the selected type of BC will be available.</p>
Variables	Variables used in this boundary condition.
Velocity	Parameters for Velocity variable.
Velocity at inf.	<p>Velocity components at infinity along axes X, Y, Z, [m/s]</p> $V_{\infty} = V_{user}$
X	The component of velocity at infinity along the X axis

Y	The component of velocity at infinity along the Y axis
Z	The component of velocity at infinity along the Z axis
Pressure at inf.	User-defined relative pressure value at infinity, [Pa] $P_{\infty} = P_{user}$
Temperature	Parameters for Temperature variable.
Value	User-defined relative temperature value at infinity, [K] $T_{\infty} = T_{user}$

5.4.6.6 External BC

External BC boundary condition automatically appears in the project if the project is created by **External flow template**, and sets the type of surface for the external box around the object.



All context menu commands are not available for **External BC**, because this type of boundary condition is unique and can be present in the project in the singular.

Also, the display parameters and parameters that affect variables (except for **Turbulence level**) are not available for **External BC**. Calculations in this boundary condition are performed according to strictly defined equations, depending on the type.

Parameters

Name	The name of the Boundary Condition that is displayed in the project tree.
Variables	Variables used in this boundary condition.
External BC type	Boundary condition type for external box: Inlet/Outlet is the arrangement of the Inlet/Outlet type boundary conditions for the flow around the object according to the motion vector of the medium specified in the initial conditions. Non-reflective - all surfaces of the outer box will have the Non-reflective type boundary conditions.
Turbulence level	The conditional level of turbulence. Available if the External BC type = Inlet/Outlet. Possible values: Low, 3% Medium, 10% High, 20% .

5.4.6.7 Source of moisture

Context menu commands

Remove BC	Delete the selected Boundary condition #N.
Copy	Copy the selected Boundary condition #N.
Create Result	Create Result for the selected Boundary condition #N.

Parameters

Name	The name of the Boundary Condition that is displayed in the project tree.
Color	The color for displaying the Boundary condition in the graphical area.

	You can select a color from the drop-down list of standard colors or select a custom color using the Color = Other...
Transparency	Transparency adjustment by a fractional value in the range from 0 to 1.
Type BC	Type of boundary condition. When you change the type of boundary condition, available parameters in properties window change. Only the parameters that can be set for the selected type of BC will be available.
Variables	Variables used in this boundary condition.
Temperature	Temperature parameters for source of moisture.
Method of setting temperature	The method of setting the flow temperature from the source. Possible options: <ul style="list-style-type: none"> • Air conditioner cooling is a relative value defined as the difference between the temperature of the region and the temperature of boundary condition. • Constant is a fixed specified value.
Value	Specified value of temperature, [K]
Velocity	Parameters for Velocity variable.
Method	The method of setting Velocity variable. Possible options: <ul style="list-style-type: none"> • Inlet pressure — relative inlet pressure, [Pa]. • Fixed velocity — fixed velocity with selection of components of the flow velocity vector along the X, Y and Z axes, [m/s]. • Mass flow — the mass flow rate of the substance through the surface of the boundary condition, [kg/s]. The formulas and features for each method are described in Inlet/Outlet section.
Pressure	Relative static inlet pressure. Available if Method = Inlet pressure .
Velocity	The normal flow rate, [m/s]. Available if Method = Fixed velocity .
X	Components of the flow velocity vector along the X, Y, and Z axes, [m/s].
Y	
Z	
Mass flow	Mass flow through the boundary condition, [kg/s]. Available if Method = Mass flow .
Humidity	Air humidity parameters.
Method for specifying the mass fraction of water	Method for calculating the mass fraction of water vapor in the air. Possible options: <ul style="list-style-type: none"> • Mass fraction; • Relative humidity, calculated by formulas: $d_{vap} = \varphi d_s ;$ $Y_{vap} = \frac{\varphi d_s}{1 + \varphi d_s} .$ • Specific humidity, calculated by formulas: $d_{vap} = m_{vap} / m_{air} ; Y_{vap} = m_{vap} / (m_{vap} + m_{air}) ;$ $\frac{1}{Y_{vap}} = \frac{m_{vap} + m_{air}}{m_{vap}} = \frac{1}{d_{vap}} + 1 = \frac{1 + d_{vap}}{d_{vap}} ;$

$$Y_{vap} = \frac{d_{vap}}{1 + d_{vap}}.$$

Value	The specified value of the water vapor content in the air according to the selected method.
-------	---

Notations for formulas

p — mixture pressure;

p_{vap} — partial pressure of water vapor;

p_s — saturation pressure;

d — mass moisture content;

φ — relative humidity;

x_i — molar fraction of substance i ;

Y_i — mass fraction of substance i .

5.4.6.8 Hood

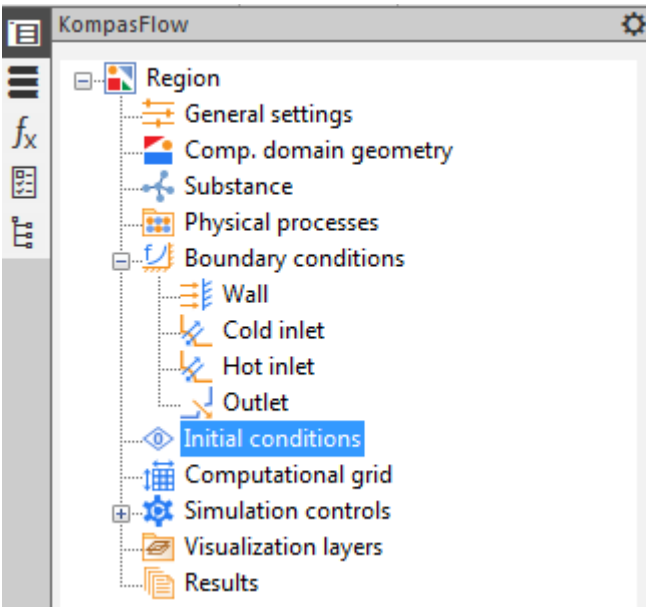


The parameters that affect variables are not available for **Hood** boundary condition. Calculations in this BC are performed according to strictly defined equations, similar to those for BC Free outlet.

Context menu commands

Remove BC	Delete the selected Boundary condition #N.
Copy	Copy the selected Boundary condition #N.
Create Result	Create Result for the selected Boundary condition #N.

5.4.7 Initial conditions



The Initial conditions element sets initial conditions of simulating.

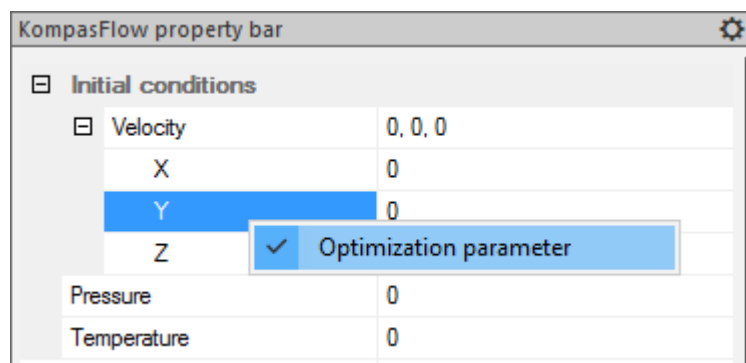
The Initial conditions element has no context menu.

Velocity	
X	The component of velocity along the X-axis [m/s]
Y	The component of velocity along the Y-axis [m/s]
Z	The component of velocity along the Z-axis [m/s]
Pressure	Pressure value [Pa]
Turbulence level	<div>This is an indicative level of turbulence. Available options: Low, 3% Medium, 10% High, 20%. This parameter is available only when Equation of motion = Yes and Turbulence = Yes is set Physical processes.</div>
Temperature	Temperature value [K]
Humidity	
Method for setting the mass fraction of water	<div>A method for setting the mass fraction of water for modeling water vapor in the air. Possible values: Relative humidity The formulas for calculating the methods can be found in Source of moisture section for Humidity > Method for setting the mass fraction of water.</div>
Value	The value of the water vapor content in the air according to the selected method.

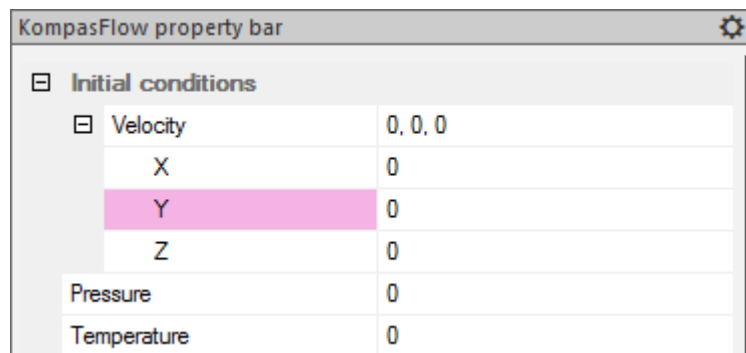
Declaring a numerical property of Initial conditions as an optimization parameter

Numerical parameters of Initial conditions can be declared as [optimization parameters](#).

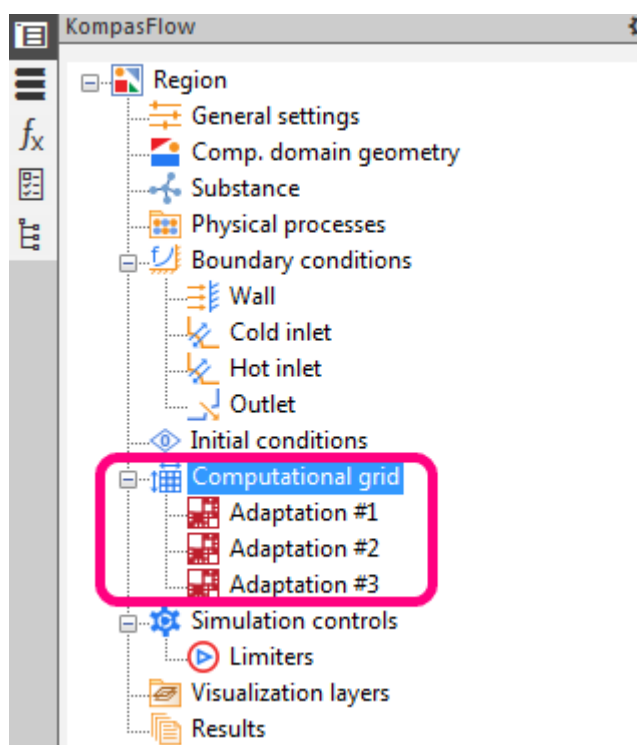
To do so, you have to open the context menu of a parameter and enable the Optimization parameter option:



Optimization parameters are highlighted in the properties of Initial conditions with pink color:



5.4.8 Computational grid and adaptation



The Computational grid folder and its child elements Adaptation #N set the properties of the computational grid and its adaptations.



Adaptations can be created, deleted, enabled or disabled in connection to the solver (the calculation process is to be suspended).

Disabling or deleting an Adaptation causes merging its earlier refined cells.

Disabling the Adaptation to solution also causes merging its earlier refined cells.

Context menu commands

Computational grid folder


Create adaptation	This command creates a new element Adaptation #N.
	You can also create an adaptation by clicking  icon in the Task statement command group of the KompasFlow toolbar .

Adaptation #N item

Delete	This command deletes the selected Adaptation #N.
--------	--

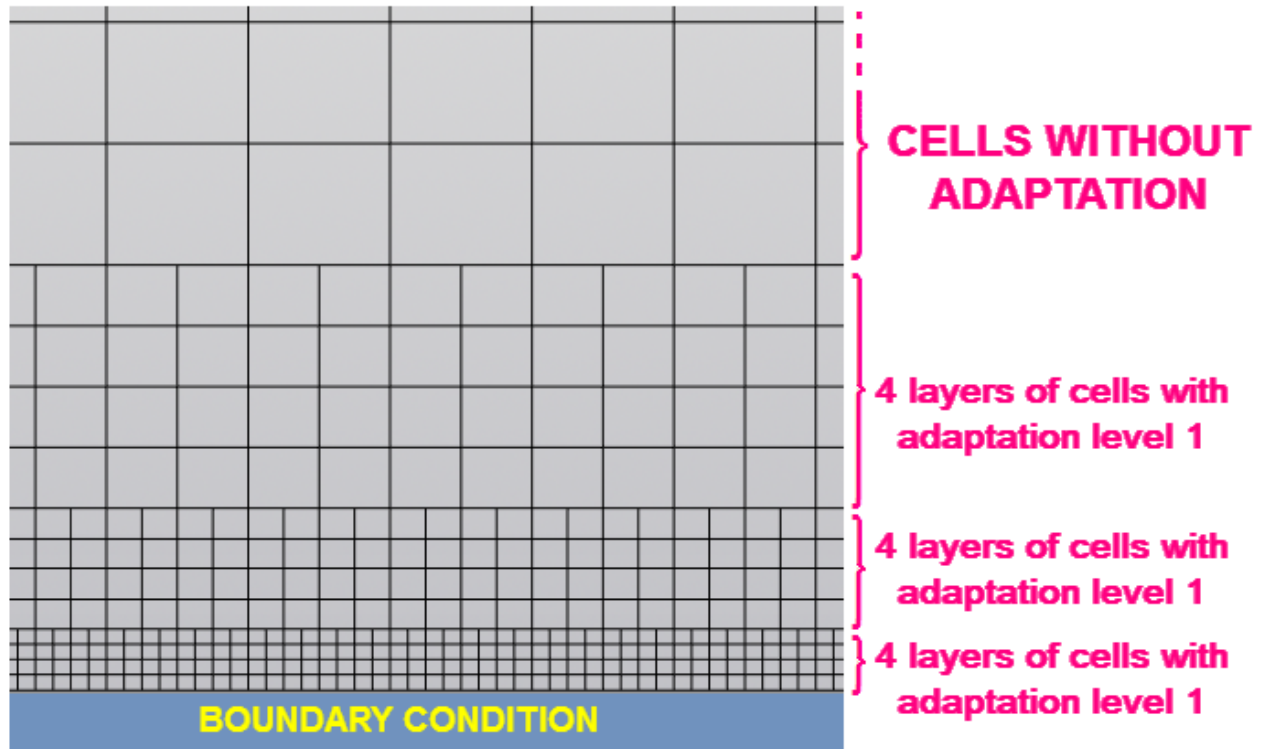
Parameters

Computational grid folder

Initial grid	
Grid setting method	The method of initial grid construction: Number of axis splits — the grid dimensions are set by the number of cells along the coordinate axes. Total number of cells — the grid dimensions are set by the total number of cells.
nX	The number of cells of the initial grid along the X axis (available if Grid setting method = Number of axis splits)
nY	The number of cells of the initial grid along the Y axis (available if Grid setting method = Number of axis splits)
nZ	The number of cells of the initial grid along the Z axis (available if Grid setting method = Number of axis splits)
Number of cells	(available if Grid setting method = Total number of cells)
Partitioning mode	(available if Grid setting method = Total number of cells)
Adaptation to solution	Applying an adaptation to solution. Possible options are: Yes No.
	All of the following options are available only if Adaptation to solution = Yes.
Start	An iteration, starting from which the adaptation to solution will be active.
Duration	How many iterations the adaptation to solution will be active (within one period)
Period	Frequency of starting the adaptation to solution
Variable	A variable that triggers applying the adaptation to solution. Possible options: Pressure Temperature Density Velocity.
Adaptation to gradient	Apply the adaptation to solution depending on the gradient of the selected Variable
Value	Target value for the adaptation to solution. Adaptation to solution will be applied when values of the Variable are close to the specified here Value.
Max amount cells	The maximum quantity of cells used for the adaptation to solution

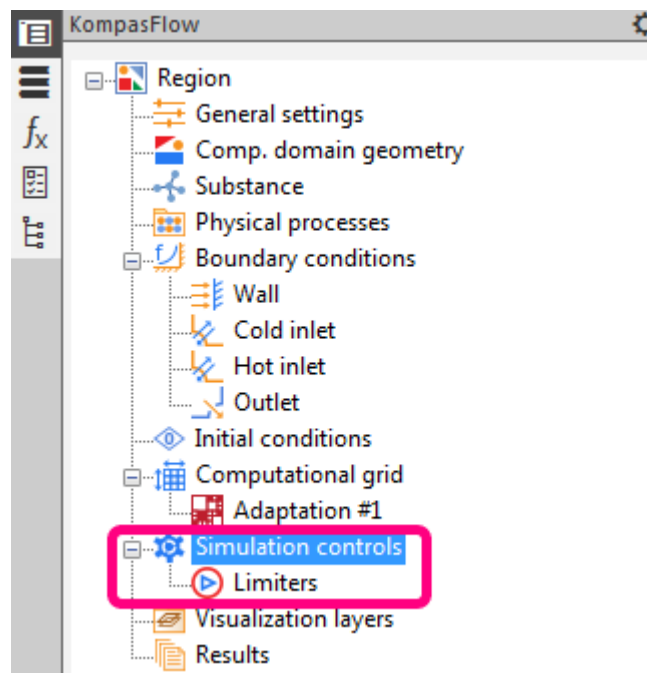
Max level	The maximum level of the adaptation to solution
Adaptation #N item	
Name	By setting the value of this parameter, you can change the element's default name Adaptation #N
Active	This parameter defines whether this Adaptation will be active (enabled) or not active (disabled). Available options: Yes No.
BC	Boundary condition on which Adaptation #N will be applied
Level	The maximum adaptation level allowed for Adaptation #N
Number of layers	Number of layers for each level of adaptation. This parameter has to be no less than 3. If you try to set a value less than 3, the program will display the message "A quality solution requires a minimum of 3 layers between the levels of grid adaptation". In rare cases, due to features of the geometry, the quantity of layers in the generated grid can differ from the specified value. It is a normal situation and not an error.
Adaptation to curvature	
Enable	Automatic increasing the maximal adaptation level near curvatures. Possible options are: Yes No.
Add. max. level	Addition to the maximal level of adaptation (which is set by the Level parameter, see above) that is added when adaptation to curvature activates. This parameter is available when Adaptation to curvature > Enable = Yes.
Max. angle	This is the threshold value of the angle between normals to facets, which, when it is exceeded, causes activation of adaptation to curvature. This parameter is available when Adaptation to curvature > Enable = Yes.
Upper limit	The upper limit for the angle of normal spreading, above which the adaptation to curvature is not applied. This parameter is available when Adaptation to curvature > Enable = Yes.
Adaptation to sharp edges	
Enable	Automatic increasing the maximal adaptation level near sharp edges. Possible options are: Yes No.
Add. max. level	Addition to the maximal level of adaptation (which is set by the Level parameter, see above) that is added when adaptation to sharp edges activates. This parameter is available when Adaptation to sharp edges > Enable = Yes.
Sharp edge angle	This is the threshold value of the angle between normals to adjacent facets (either facets that contact the boundary between groups or those ones that belong to the same group); when this value is exceeded, the adaptation to sharp edges is applied. This parameter is available when Adaptation to sharp edges > Enable = Yes.

Illustration



Layers of cells with various levels of adaptation (4 layers for each level).
Adaptation parameters are set: Level = 3, Number of layers = 4.

5.4.9 Computation parameters and limiters



The Simulation controls element and its child element Limiters set parameters of the calculation.



Limiters are most commonly used to smooth negative numerical effects when calculating supersonic flows.

These elements have no context menus.

Parameters

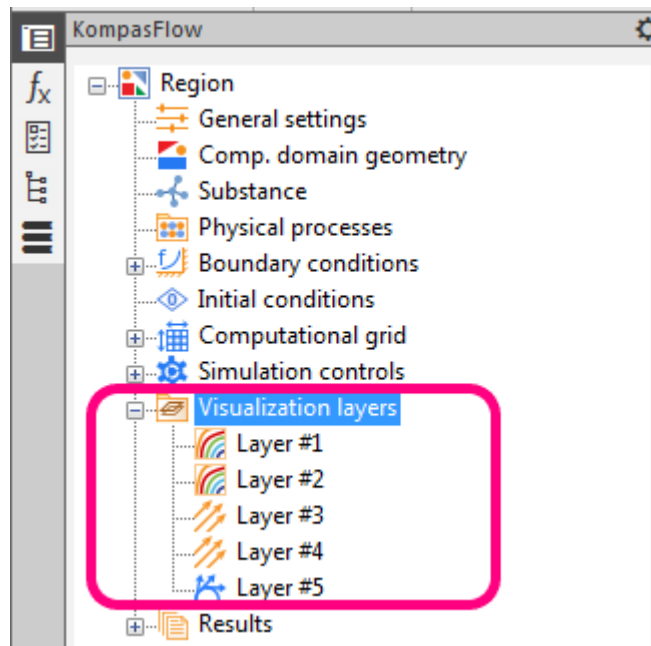
Simulations controls:

Method	Selection of the method of specifying the time step. Possible options: Via CFL number In seconds.
CFL	The Courant-Friedrichs-Lewy number. A CFL number represents the ratio of the time step to the time for which a perturbation of the flow is transferred by the stream within a cell. This parameter is only available when Method = Via CFL number. You can read more information about selecting the CFL number for various problem settings in this article .
Constant step	A constant time step specified in seconds. This parameter is only available when Method = In seconds.
Max time step	Maximal time step, [s]. This parameter is only available when Method = Via CFL number.
Simulated time	Simulated time, [s]. This parameter is only available when Method = Via CFL number.
Save frequency	Number of iterations, over which the solver will regularly save data.

Limiters:

Density, min.	Minimum density, [kg/m ³]
Velocity, max.	Maximum velocity, [m/s]
Pressure abs., min.	Minimum value of absolute pressure, [Pa]. This value can only be positive. Incompressible liquid can have any pressure.
Pressure abs., max.	Maximum value of absolute pressure, [Pa]
Nu turb., min.	Minimum admissible turbulent kinematic viscosity, [m ² /s]
Nu turb. / Nu mol., max.	Maximum admissible turbulent kinematic and molecular viscosity [m ² /s]
Temperature abs., min.	Minimum value of absolute temperature, [K]. This value can be positive only.
Temperature abs., max.	Maximum value of absolute temperature, [K].




5.4.10 Visualization layers







The Visualization layers folder contains child elements Layer #N that represent layers for visualization a flow, a variable field, or cross-section of the computational grid.

The Visualization Layers folder has no properties in the properties panel.






Context menu commands

Create layer	<p>Create a new visualization Layer #N.</p> <p>The program will prompt you to set some parameters for the new Layer and the object on which the Layer will be built. These parameters are set in the Parameters tab in the control panels area of KOMPAS-3D. To define the geometrical objects on which the new Layer will be built, it is necessary to open the Tree tab, or display a tree of geometrical model in graphical area by clicking the icon .</p> <p>After all necessary parameters have been set, the icon  will appear. Click it to complete a layer creation.</p> <p>The icon  will not appear until all required parameters are set.</p>
--------------	--



You can also create a Layer by clicking an appropriate icon (, , , ) in the Visualization command set in the [KompasFlow toolbar](#).

Types of layers

	Color contours	Display the field of the selected variable as a surface (on a plane or on a boundary condition) colored depending on the variable value.
	Vectors	Displays the field of the selected vector variable in as arrows that can be colored depending on the value of the same or another variable.
	Streamlines	Display the lines of the flow that built by the selected vector variable; streamlines can be colored depending on the value of the same or another variable.
	Computation grid section	Displays a cross-section of the computational grid by a Plane.
	Isosurface	Displays surfaces in space on which the specified variable takes on a constant value.

See details

[General properties of layers](#)

[Color contours](#)

[Vectors](#)






[Streamlines](#)

[Computation grid section](#)




[Isosurface](#)

5.4.10.1 General properties of layers


Types of layers

	Color contours	Display the field of the selected variable as a surface (on a plane or on a boundary condition) colored depending on the variable value.
	Vectors	Displays the field of the selected vector variable in as arrows that can be colored depending on the value of the same or another variable.
	Streamlines	Display the lines of the flow that built by the selected vector variable; streamlines can be colored depending on the value of the same or another variable.
	Computation grid section	Displays a cross-section of the computational grid by a Plane.
	Isosurface	Displays surfaces in space on which the specified variable takes on a constant value.

Context menu of Layer #N

Delete	Delete the selected Layer #N
Copy	Copy the selected Layer #N
Show legend	<p>Enable displaying a legend for the selected Layer #N. When the legend is displayed, the command is marked with the  sign.</p> <p>If you wish, you can move the legend to another location within the program's window. To do so, click the legend's field or header and, holding the left mouse button, move the legend to another location. The mouse cursor during this move looks like a hand:</p>  <p>You can create a legend individually for each of multiply Layers and locate the legends arbitrary in the program's window.</p>
Load palette	Load a palette for Layer #N from a file
Save palette	Save the Layer #N's palette in a file
Hide	<p>Switch off or on displaying the selected Layer #N in the graphics area of the KOMPAS-3D window.</p> <p>If a Layer is already hidden, the command is marked with the  sign.</p> <p>Icons of hidden Layers are displayed in the project tree in faded colors.</p>

Parameters (when a layer is creating)

Name	Name of the Layer. Instead of the standard name Layer #N, you can specify another name, for example, Temperature in horizontal section.
Layer type	Layer type. The available types are listed above.
Object type	This parameter indicates the type of the object, on which the Layer is built. Available options: Plane Surface Comp. space.
Emitter object type	The type of object for Streamlines source. Possible options: Plane Surface. Available only for Streamlines layer if Object type = Comp. space.
Plane / Emitter plane	The selected plane or surface on which the source of the streamlines will be set. Available only for Streamlines layer if Object type = Comp. space.
Plane	The plane on which the Layer is built. This parameter is available when Object = Plane. It is possible to select another Plane from the drop-down list.
Variable	<p>The variable by which the layer is built. The list of available variables depends on the selected layer type. All variables are listed below:</p> <ul style="list-style-type: none"> • Velocity, [m/s] • Density, [kg/m³] • Pressure, [Pa] • Temperature, [K] • Mach Number • Total temperature, [K] • Total pressure, [Pa] • Specific humidity (only for Humidity template) • Absolute humidity (only for Humidity template) • Relative humidity (only for Humidity template) • Partial pressure (only for Humidity template)
	 Vectors and Streamline layers are created by Velocity variable only. The variable selection is not available for Computation grid section layer.
Direction variable	The direction of lines (only for Streamlines layer). Options: Forward Backward Both ways .
Color variable	The variable for coloring a layer (only for Vectors и Streamlines layers). The list of variables for coloring is the same, as the list of variables for creation a layer.

Additional information for variables

- For the variables Pressure and Temperature, the excess of the absolute value over the [reference value](#) is visualized.
- When visualizing the Pressure variable, static pressure values are shown without accounting the hydrostatic component.
- If Layer type = Vectors or Layer Type = Streamlines, then the Layer can only be built using variable Velocity.

If Layer type = Fill, then it is possible to select a variable: Velocity | Density | Pressure | Temperature | Mach number | Total pressure | Total temperature | Y+ | Radiation density.
- Mach number can be visualized by the Color contours layers, as well as used as an additional variable for filling Vectors and Streamlines.

Mach number can not be set for layers built on the Wall-type boundary conditions.

5. The variables Specific humidity, Absolute humidity, Relative humidity and Partial pressure are only available for the [Humidity](#) template.
6. The variable selection is not available for the Calculation Grid Section layer.
7. Y+ can be visualized only by the Color contours layers built on Wall type boundary conditions if the physical turbulence process is activated ([Physical processes](#) > Turbulence = Yes).
8. Variable Radiation density is available if the physical radiation process is activated ([Physical processes](#) > Radiant transfer = Yes).

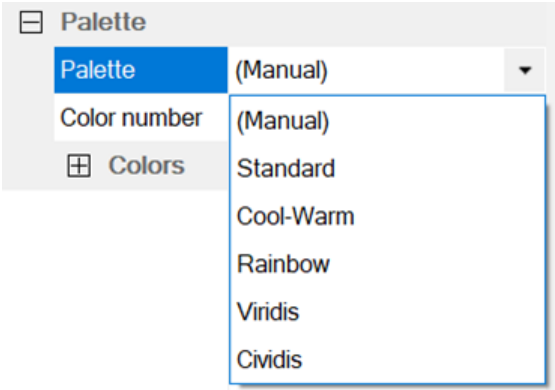
Parameters

The parameters depend on type of layers. The table below lists all the available parameters.

Name	Name of the layer. Instead of the standard name Layer #N, you can specify another name, for example, Temperature in horizontal section.
Layer type	The type of layer is specified when a layer is created.
Object	The object on which the layer is built. The object is specified when a layer is created.
Plane	The plane on which the Layer is built. This parameter is available when Object = Plane. It is possible to select another Plane from the drop-down list.
Surface	The boundary condition on which the Layer is built. Available if Object = Surface. The surface is specified when a layer is created.
Variable	The variable by which the layer is built. The list of variables is listed above.
View settings	
Visible	This parameter specifies if the Layer is displayed in the graphical area of the KOMPAS-3D window. This parameter can be changed by the context menu command Hide. Possible options are: Yes No.
Show legend	Displaying a legend (explanatory label) in the graphical area of the KOMPAS-3D window. This parameter can be changed using Show legend command in the context menu. The legend scale is divided into 10 intervals and has 11 figures with numbers. Possible options: Yes No.
Legend parameters	
Style	Orientation of the legend. Possible options are: Vertical Horizontal.
Value	Format of numbers in the legend. Possible options are: Auto Fixed Scientific.
Precision	This parameter defines how many valid digits will be displayed in numbers in the legend.
Frame	This parameter defines if the legend will be displayed with a frame. Possible options are: Yes No.
Transparency	Transparency of the legend's background specified as percentage by a number in the range from 0 to 100
Range	Settings of the range for displaying the main visualized Variable (for a Color contours layer) or the Color variable (for layers Vectors and Streamlines).

Mode	Range selection mode. Available options: <ul style="list-style-type: none">• Local: the range spreads from the minimum to the maximum values within the Layer.• Global: the range spreads from the minimum to the maximum values within the entire computational domain.• Manual: the range is set manually by the user.
Maximum	Maximum range value
Minimum	Minimum range value
Palette	Group of parameters for palette configuration.

Palette	Select a ready-made (preset) palette or a custom (manually defined) palette. The following ready-made palettes are available: Standard Cool-Warm Rainbow Viridis Cividis.
---------	--



To use a custom palette, select View settings > Palette > Palette = (Manual).
If the number of colors or any color of the palette has been changed, the palette becomes manually configured and available for later use when selecting Display Settings > Palette > Palette = (Manual), see above.
Custom palette settings are set by parameters below.

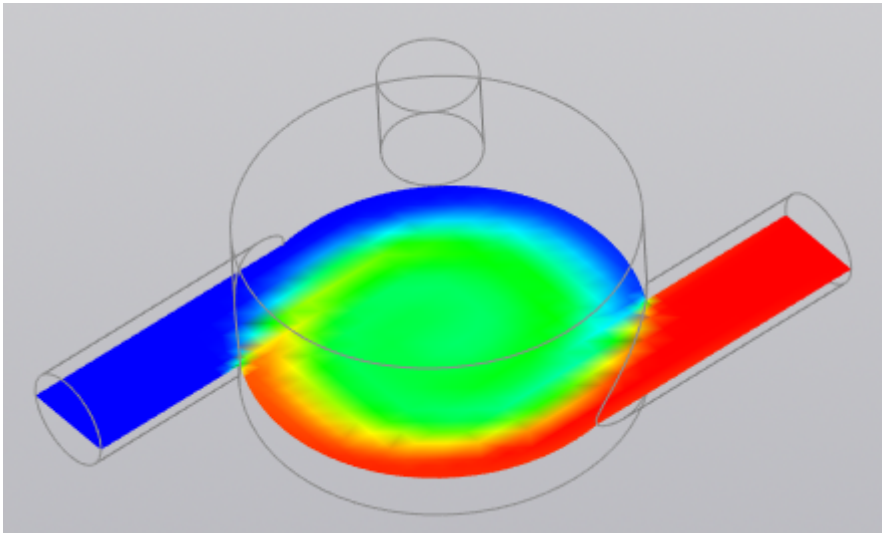
Color number	The number of colors in the palette used to visualize the variable. It is set as value in a range from 2 to 21.
Colors	If you need to change a color in the palette, click its line in the table, and then click the "▼" symbol; after this specify the color in the standard operating system dialog box for selecting colors will open.

Colors		
70		Red
65.3571		Other...
60.7143		Other...
56.0714		Other...
51.4286		Other...
46.7857		Other...
42.1429		Other...
37.5		Lime
32.8571		Other...
28.2143		Other...
23.5714		Other...
18.9286		Other...
14.2857		Other...
9.64286		Other...
5		Blue

If number of colors or any color in a palette were changed, this palette becomes the non-standard (manual) one and it also will be available for future use when you select View settings > Palette > Palette = (Manual), see above.

Drawing method	The method of displaying the layer in the view window. Possible options: Color fill Color fill with separation of levels.
----------------	---

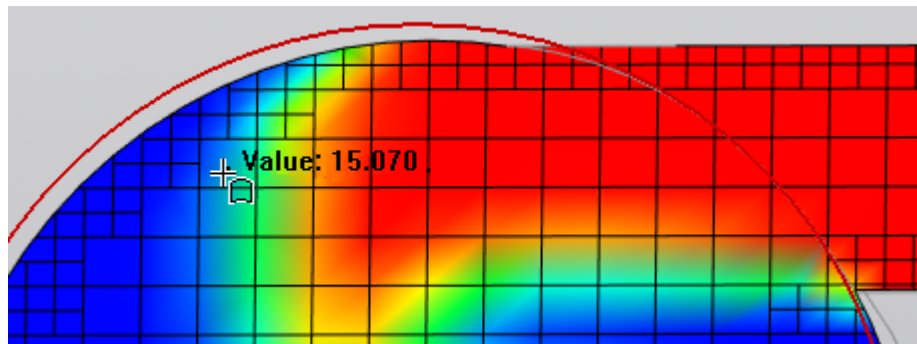
5.4.10.2 Color contours



The Color contours layer has no specific parameters.

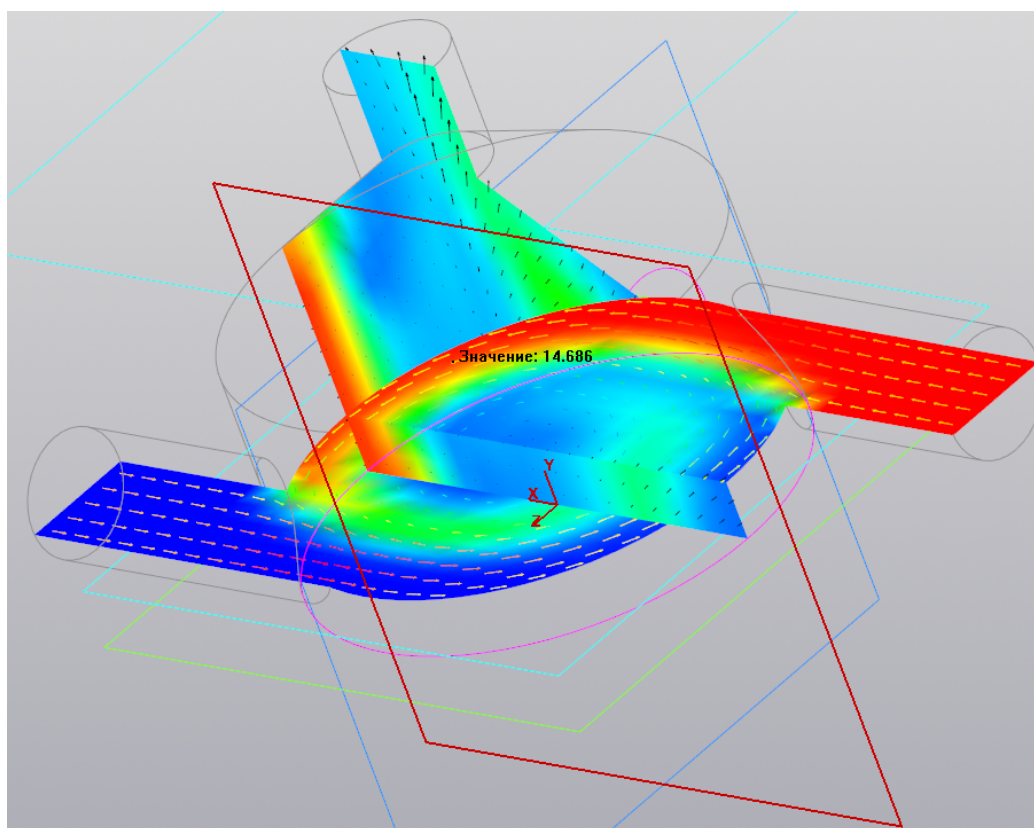
Additional information

When the Color contours layer is selected in the project tree while and the Alt key is pressed, near the mouse cursor in the graphical area the value of the visualized variable will be displayed (as a text line) at the point of intersection of the line going from the observer to the mouse cursor with the Layer surface.

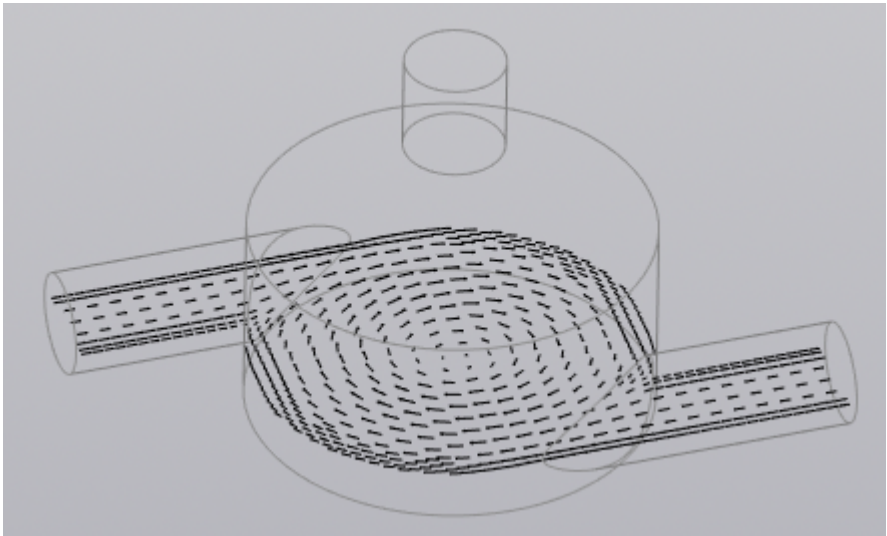


Regardless of the number of non-hidden Color contours layers, the numeric value will be shown only for the Color contours, which is selected in the project tree.

Example: The illustration below shows how a numerical value is displayed for the layer selected in the project tree and built on a plane marked with a red rectangular contour, and not for the layer that locates closer to the observer. (The image is given in the view from the bottom of the mixer.)



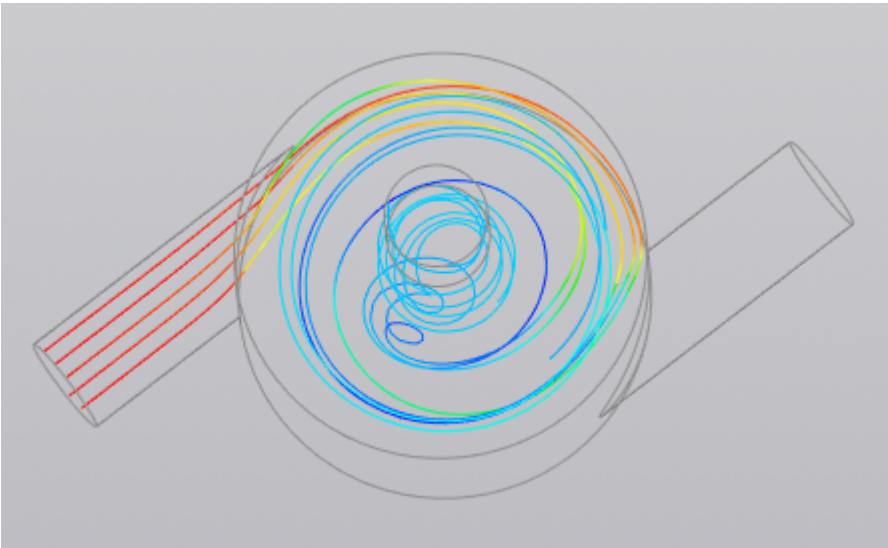
5.4.10.3 Vectors



Specific parameters	
Color variable	A variable that is used for coloring images of the vectors. Available options: No Velocity Density Pressure Temperature Mach number Total pressure Total Temperature Radiation density.
View settings	
Constant length	Draw all vectors with constant length. Available options: Yes No.
Reference length	The reference vector length, [mm]
Density	Density of displaying the vectors (a relative value specified in percent)

5.4.10.4 Streamlines

Streamlines can be drawn either in Computational space or on a Plane or on a Surface. This is set by selection of the Object parameter.

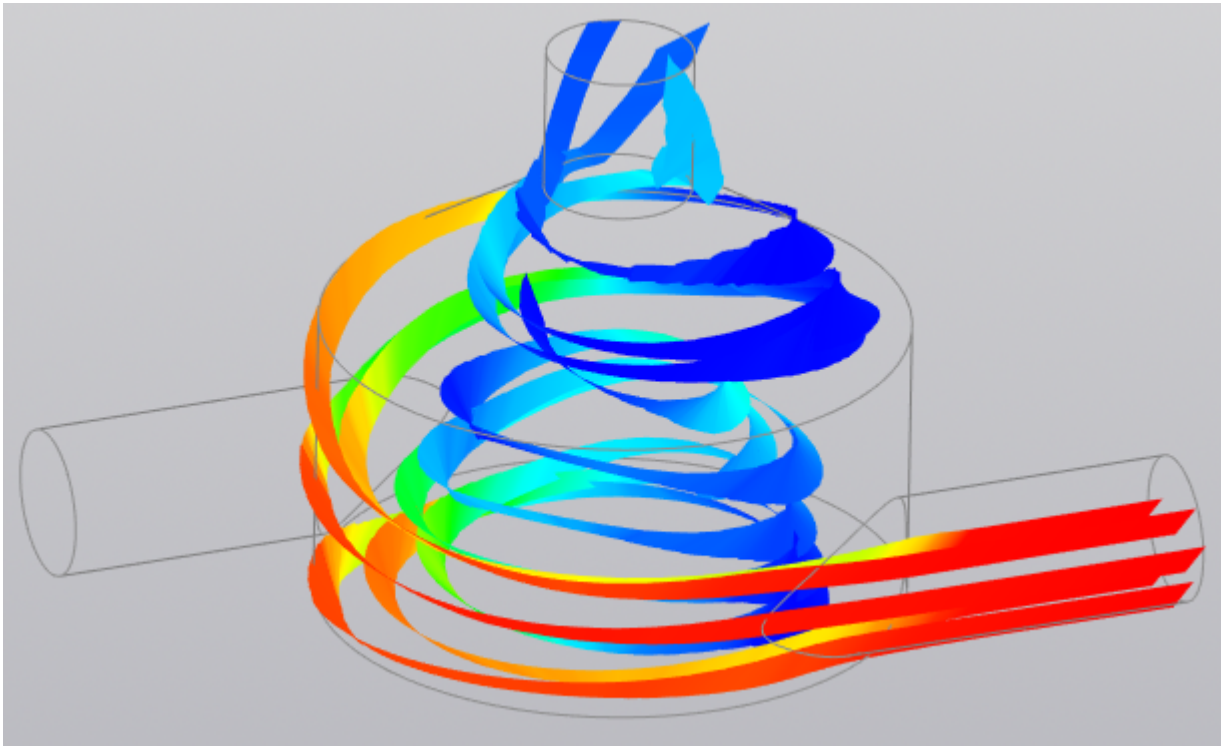


Standard displaying streamlines (Drawing mode = Lines)

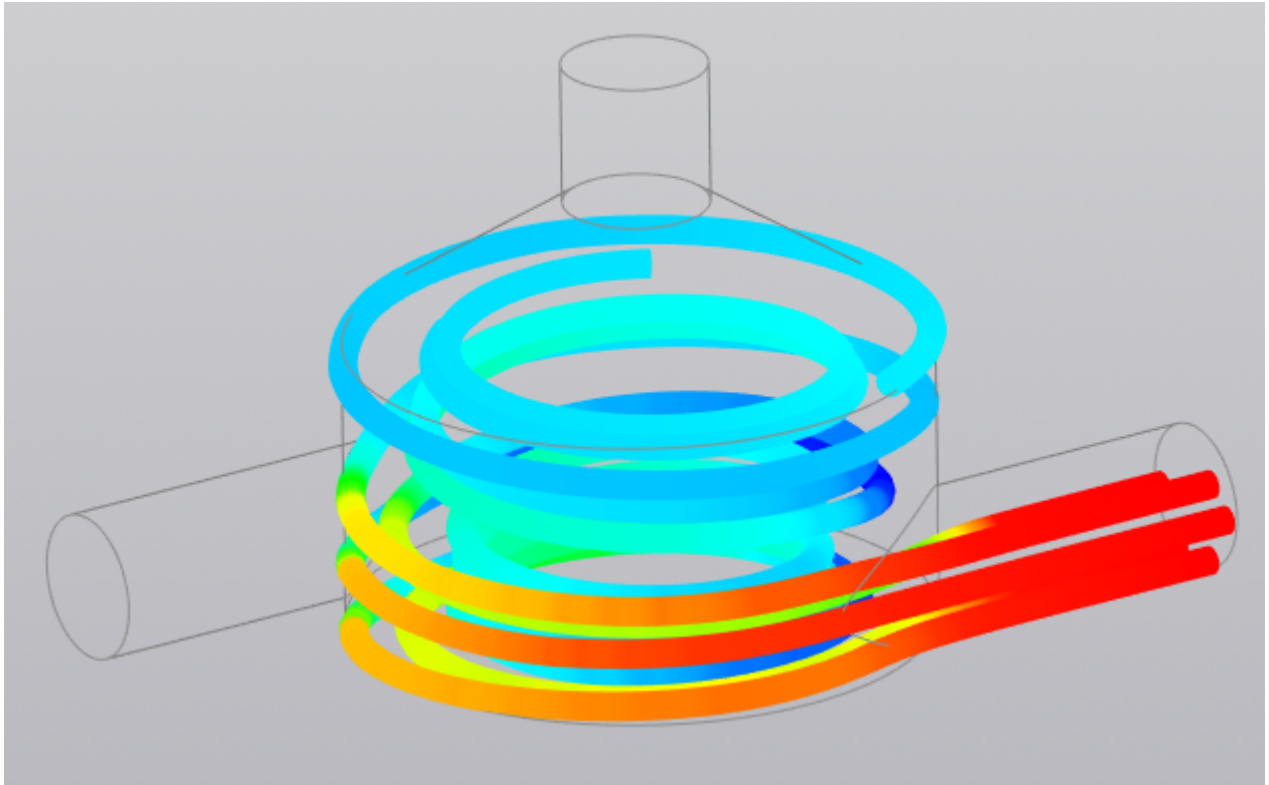
Parameters

Color variable	<p>A variable that is used for coloring images of the streamlines.</p> <p>Available options: No Velocity Density Pressure Temperature Mach number Total pressure Total temperature Radiation density.</p>
Emitter Parameters	
Max. point number	<p>The number of emitters of the streamlines.</p> <p>When this parameter is set as -1, the streamline emitters will be formed in nodes of the computational grid.</p>
Direction	<p>The direction in which the streamlines will be built from emitters.</p> <p>Available options: Forward Back In both directions.</p>
View settings	
Drawing mode	<p>How the streamlines are displayed (see illustrations).</p> <p>Available options: Lines Tapes Tubes.</p>
Line thickness	<p>Thickness of streamlines when they are displayed as Tapes or Tubes, [mm].</p> <p>When this parameter is set as -1, the default thickness will be applied.</p>

Illustrations

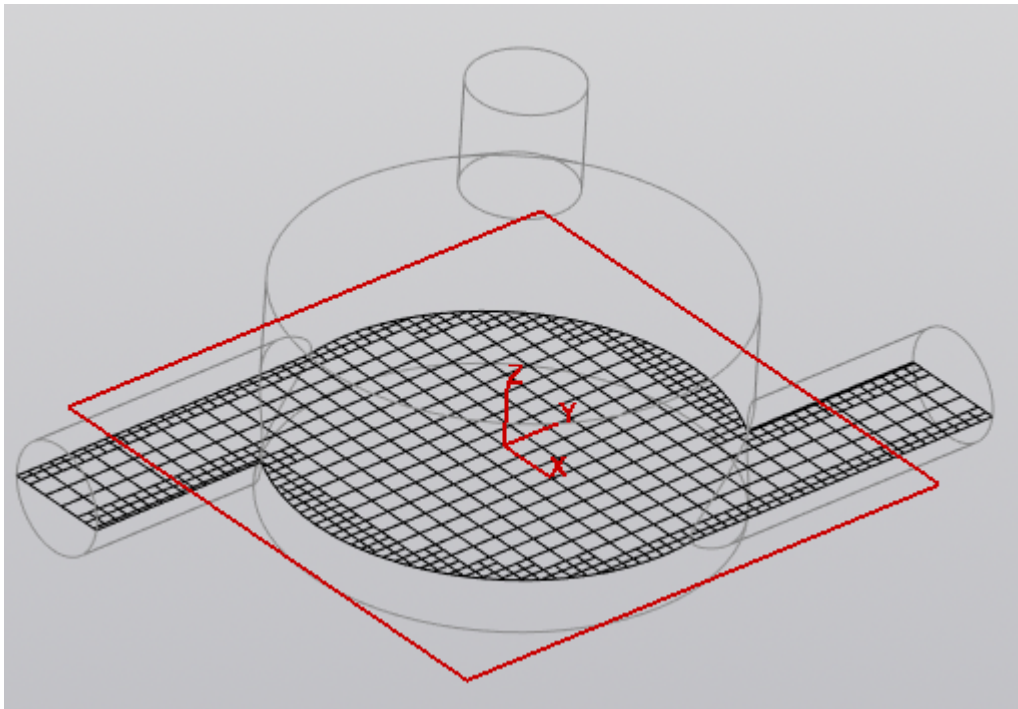


Displaying streamlines as tapes (Drawing mode = Tapes)



Displaying streamlines as tubes (Drawing mode = Tubes)

5.4.10.5 Computation grid section

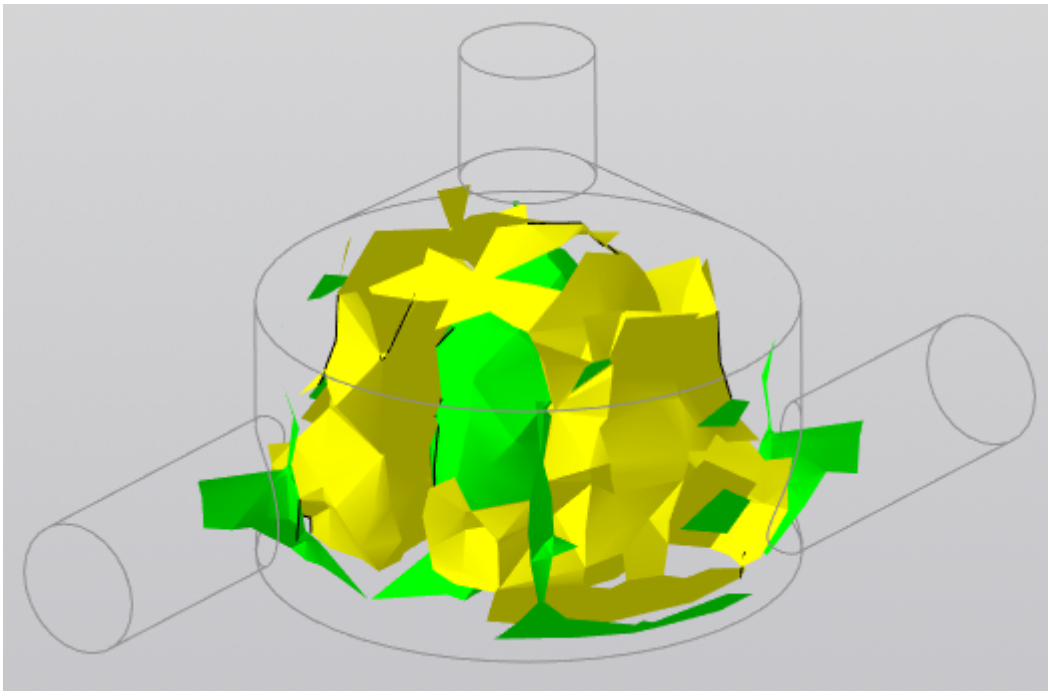


Computation grid section layer displays the section of the calculation grid along the specified plane.

5.4.10.6 Isosurface

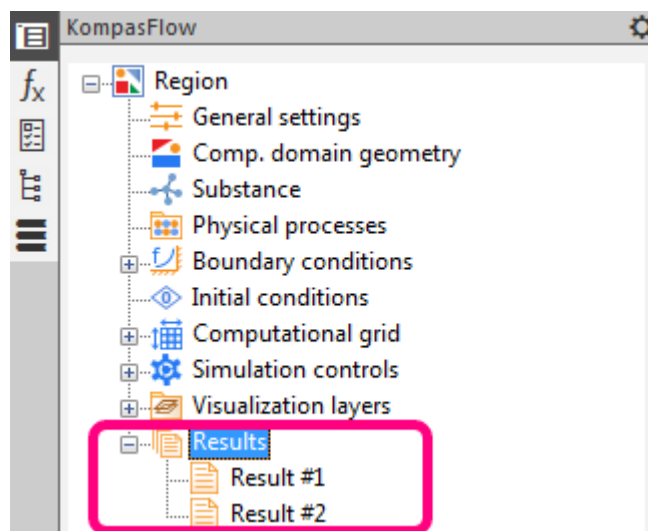
Isosurface layer displays the surfaces on which the variable takes on a constant value.

Isosurface is defined over the entire space of the calculated area and displays in one color the surfaces on which the set value of the selected variable is displayed.



The illustration above shows two isosurface layers for displaying two variable values.


5.4.11 Results (folder)





The Results folder contains child elements **Result #N** that represent the results of the computation, plots of which are displayed in the [Monitoring window](#).

Context menu














Create Result	Create a new element Result #N. The program will prompt you to set parameters of the Result to be created and the object on which it will be built. The parameters are set in the Parameters tab in the
---------------	--

KOMPAS-3D control panel area. To set the geometric objects on which the new Result will be calculated, you have open the Tree tab, or display the tree of the geometric model in the graphical area by clicking the  icon.

Having set all the necessary parameters, click the  icon that appears.



You can also create a Result:

1) click a corresponding icon (, , , , , , , , , , , , ) in the Results command set of [KompasFlow toolbar](#).

2) use context menu command Create Result of an element "Boundary conditions > Boundary condition #N"

Parameters


The Results folder has no parameters in the properties bar.

Results (child items)

See section [Results \(items\)](#).











5.4.11.1 Results (items)

Context menu

Optimization goal	Declare the selected Result #N as a goal for optimization . Icons of Results , which are declared as optimization goals, are displayed as  .
Delete	Delete the selected Result #N from the project tree.

Parameters of the element "Result #N" set at its creation

Name	Name of the Result. You can specify another name instead of the standard name Result #N.
Result	Result of the computation. Available options: <ul style="list-style-type: none">• Temperature – the excess of the absolute temperature above the reference temperature, [K]• Total temperature, [K]• Pressure – the excess of absolute pressure over the reference pressure, [Pa]• Total pressure, [Pa]• Force – the force acting on a surface, a set of surfaces or the body from the flow, [N]• Torque – the torque acting on the object (body) from the flow that is set relative to the axes passing through the origin of the object, [N·m]• Density, [kg/m3]• Velocity, [m/s]• Mach number• Heat flux – the total heat flux, [W]. Heat flux is calculated without taking into account the radiative component.• Radiation flux – radiative component of the heat flux• Mass flow, [kg/s]• Volume flow, [m³/s]

Object	Geometric object on which the Result is calculated. Available options: Comp. space Plane Surface Point.
Value	The calculated value of the Result. Available options: <ul style="list-style-type: none">  Min. means the minimum of the measured value.  Max. means the maximum of the measured value.  Average means the average of the measured value.  Std. dev. means the standard deviation of the measured value.
Direction	The direction of the flow for calculating Mass Flow and Volume Flow. Available options:  Inside  Outside
Component	This parameter makes it possible to specify either a component or the absolute value of a vector variable (calculation of the component or absolute value is carried out after averaging or integrating the vector variable). Available options:  Length  Comp. X  Comp. Y  Comp. Z

Parameters of the element "Result #N" displayed in its property bar


Name	Name of the Result. You can specify another name instead of the standard name Result #N.
Variable	The variable, based on which the Result is calculated.
Object	The object, on which the Result is calculated. This object is highlighted in the graphical area of the KOMPAS-3D window.
Value	The value, which will be calculated by the Result (for example, Min., Max., Average, Standard deviation)



Variable, Object and Value parameters are set when creating the **Result** and cannot be changed in the properties bar.

5.5 Monitoring window

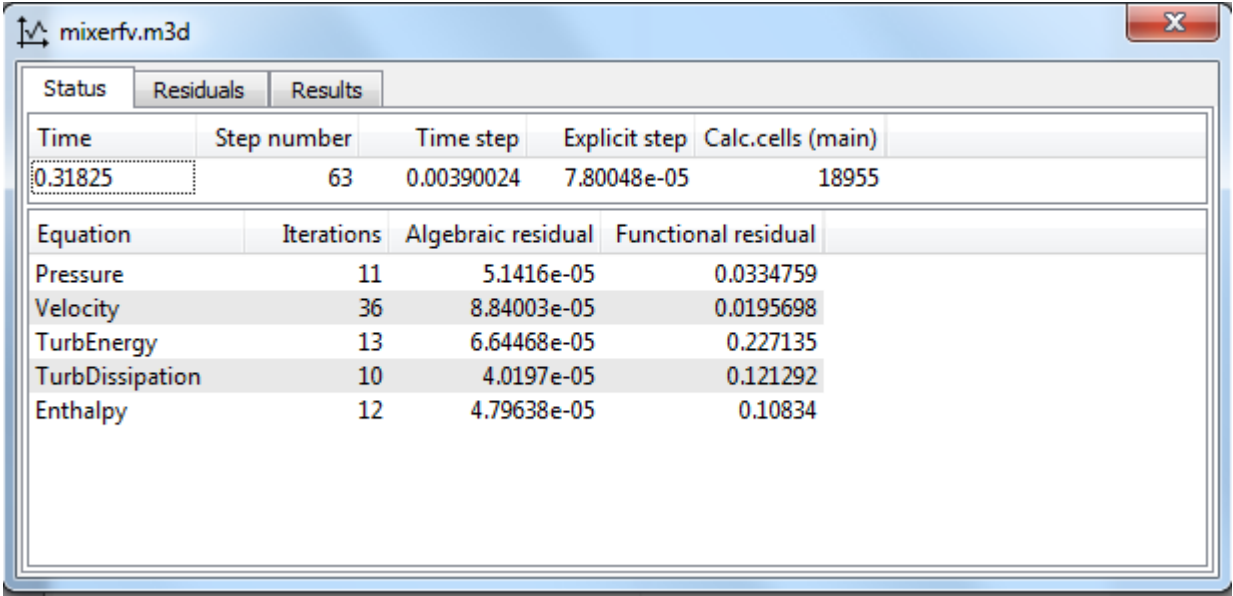
The Monitoring window contains data from the current computation. When the computation starts, this window opens in the central part of the KOMPAS-3D window.

If desired, the Monitoring window can be moved to other location or closed by clicking the x symbol in its upper right corner. To reopen the Monitoring window, click the  Open monitor icon in the [KompasFlow toolbar](#).

The Monitoring window has the following tabs:

- Status
- Residuals
- Results

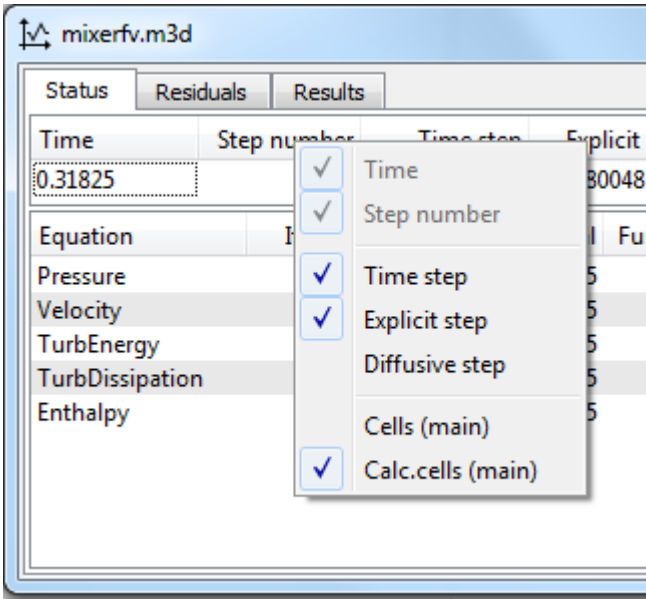
Tab "Status"



Monitoring window, the Status tab

The Status tab contains two tables.

The upper table displays the current time, the current step number, and other data whose composition is configured from the context menu that opens after a right-click on the table's header:



The lower table displays the main variables of the solved equations. Each of these variables corresponds to its own system of linear equations and a line in the table, which indicates the number of iterations made for the convergence of the solution of algebraic equations, the maximum residual over the computational domain and the functional residual.

Data from table cells can be copied using the keyboard shortcuts Ctrl+Ins and Ctrl+C.

Data displayed in the upper table of the Status tab of the Monitoring window

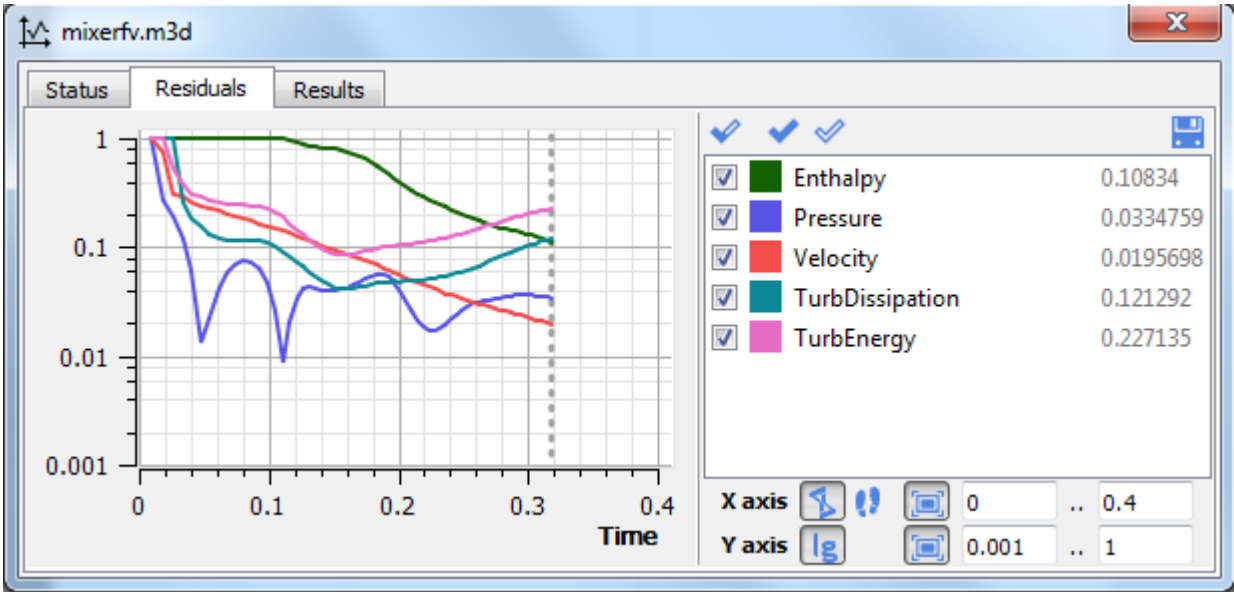
Time	Current time, T
Step number	Current time step number, n
Time step	Time step, τ

Explicit step	Explicit convective time step, $\tau_{\text{expl, conv}}$
Diffusive step	Diffusive time step, τ_{diff}
Cells (main)	Total number of grid cells
Calc. cells (main)	Number of computational cells in the grid

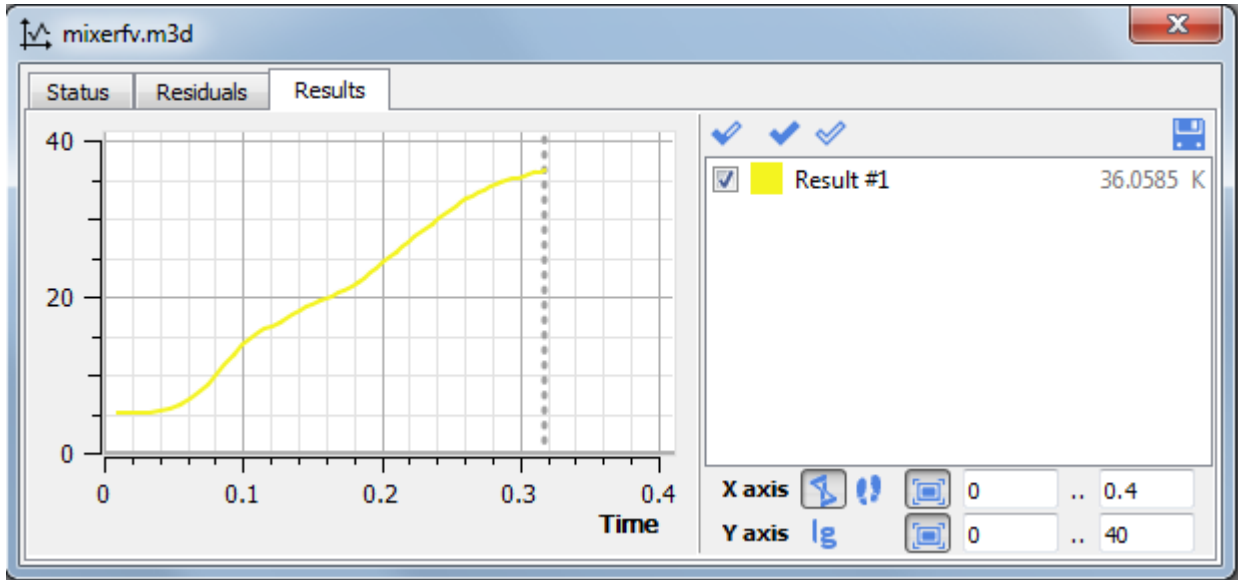
Data displayed in the lower table of the Status tab of the Monitoring window

Equation	List of main variables calculated.
Iterations	The number of iterations executed for the convergence of the algebraic equations. When a steady-state flow is calculated using the relaxation method, the number of iterations should decrease. If the number of iterations is consistently large, then the calculated process is unsteady-state.
Algebraic residual	The algebraic residual $R_A(t^n)$, this is the maximal, over the computational domain, residual obtained during solving the algebraic equations.
Functional residual	The functional residual $R_{\text{norm}}(t^n)$, this is the maximal, over the computational domain, rate of change of the main calculated variable.

Tabs "Residuals" and "Results"



Monitoring window, the Residuals tab



Monitoring window, the Results tab

On these tabs the axis of abscissa corresponds to time or to number of steps. The axial of ordinates corresponds to values of the functional residuals of the calculated variables or to the specified Results.

The current time or step is shown on the plot as a vertical dotted line.

On the right of the plot, there is a pane with a list of plot lines (with the designations of colors and values at the last step along with their dimensions) and interface controls for tuning the plot:

List of plot lines

- ☒
- ☒
- ☒
- ☒
- By checking or unchecking the check boxes, you can select the lines that will be displayed on the plot.
- Lines that are not displayed are shown in the list with a faint font. The colored square indicates the color of the line on the plot.
- Values at the last step are displayed on the right.

Bulk changes of displaying or hiding the plot lines

- ☒ Invert selection of displayed and hidden plot lines
- ☒ Display all plot lines
- ☒ Hide all plot lines

Settings for the abscissa axis (X-axis)

- Select **Time** as variable for X-axis
- Select **Step number** as variable for X-axis
- Automatically adjust the scale at the abscissa axis (so that the entire plot will fit horizontally)

0 .. 170

Fields for manual input of the plot's range of the abscissa axis. Enter the data and press the Enter key on the keyboard.

Settings for the ordinate axis (Y-axis)

- Enable/disable the logarithmic scale mode for the ordinate axis.
By default, Residuals are displayed in the logarithmic scale, while Results are displayed in the arithmetic scale.
- Automatic adjustment of the scale of the ordinate axis (so that the entire plot will fit vertically)

1e-08 .. 90

Fields for manual input of the plot's range of the ordinate axis. Enter the data and press the Enter key on the keyboard.

Saving the plot in a text file



Data from the plot can be saved in a text file, in which data columns are separated with tabulation characters.

After clicking this icon, a standard operating system's dialog box will open where you should specify the file, in which the data will be saved.

Sample of the text file:

Step	Time	Pressure	Velocity
1	100	1	
2	200	1	1
3	300	1	0.904514
4	400	0.594614	0.766632
5	500	0.0251451	1
6	600	0.132585	0.779342
7	700	0.239844	0.555458
8	800	0.351604	0.422173

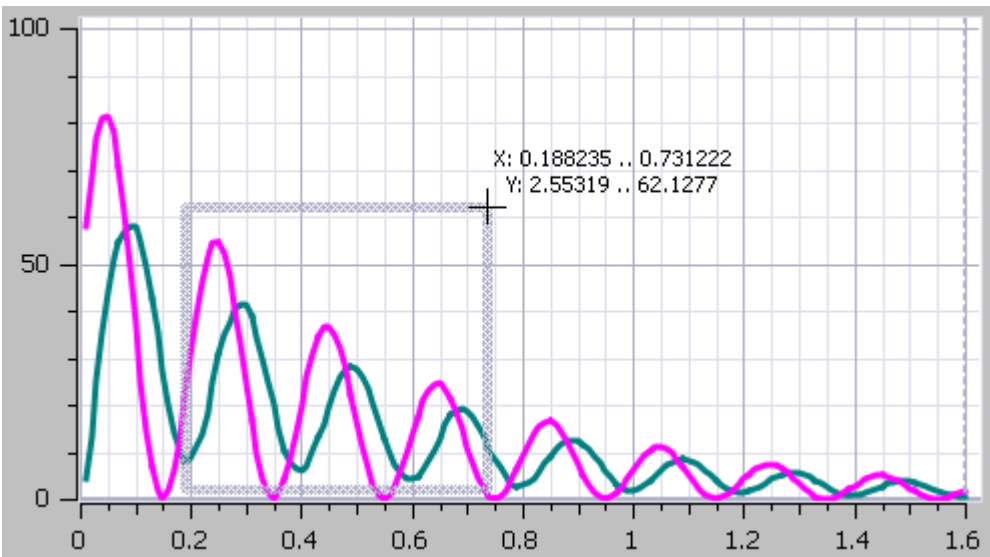
Manual scaling and/or scrolling the plot using the mouse

You can scale and/or scroll the plot manually using the mouse.

Zooming and/or shifting the graph can be done manually using the mouse. The scaling techniques are described below.

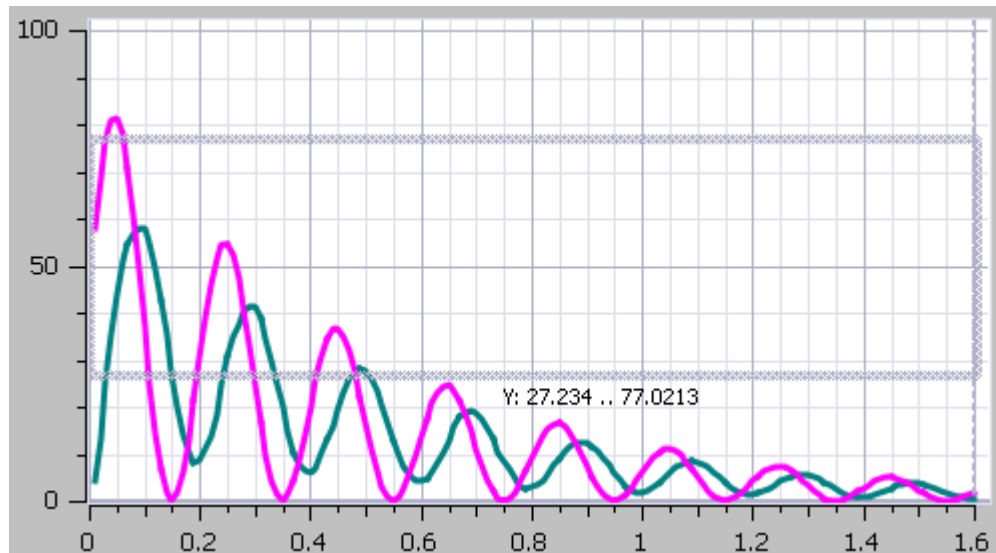
Stretch a rectangular piece of the plot across the whole pane

Left-click the mouse in the plot field and, without releasing it, move the mouse pointer to another point and then release. The selected rectangle will stretch to the whole pane of the plot.



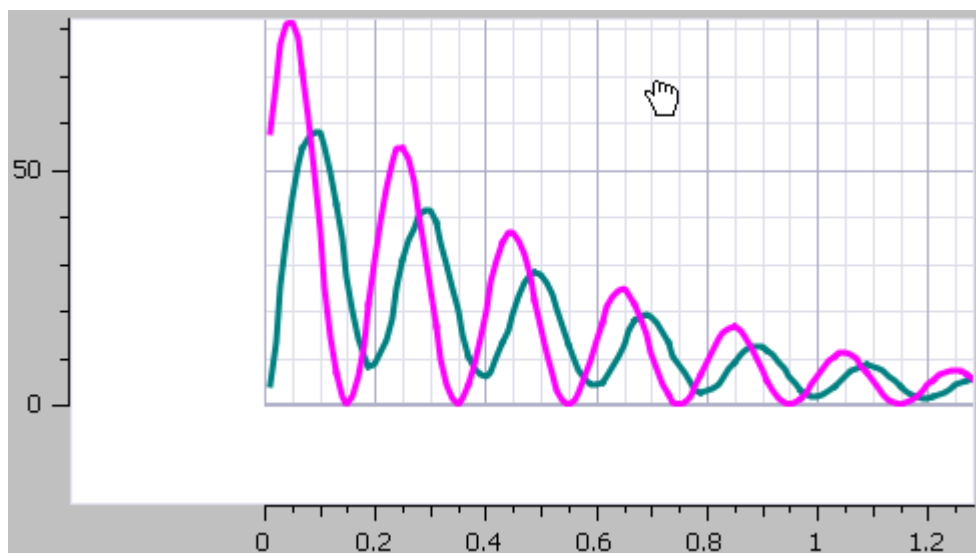
Stretch a range along one axis of the plot across the whole pane

Left-click a bit below the abscissa axis or on the left from the ordinate axis. Then, without releasing the mouse button, move the pointer along the axis to another position and release there. The selected interval will stretch to the whole pane of the plot. Next to the mouse pointer, the new range along the corresponding axis is indicated.



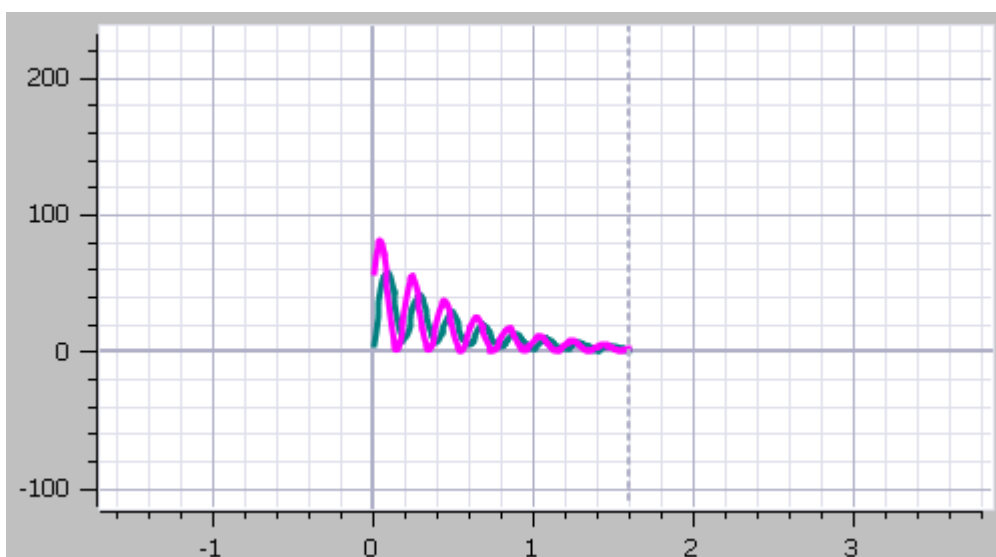
Shifting the whole plot

Right-click the mouse in the plot pane and, without releasing the button, move the plot. Scrolling that was started but not completed can be canceled by pressing the Esc key.




Scaling the whole plot


Click somewhere inside the plot pane and rotate the mouse wheel. The image will zoom in or zoom out depending on the direction of rotation of the mouse wheel.





When you move a project to a new KOMPAS-3D window (using the Move to new KOMPAS-3D window command from the context menu of the KOMPAS-3D document), the opened Monitoring window remains in the old KOMPAS-3D window. To open the Monitoring window in a new KOMPAS-3D window, close it in the old KOMPAS-3D window, navigate to the new KOMPAS-3D window, and apply the  Open monitor command from KompasFlow toolbar.

5.6 Changing a project connected to the solver

If there is a connection with the solver, most of the project parameters are not available for change by the user. Some parameters can be changed in connection with the solver; however, this requires stopping the calculation using the  Stop command. Please note that some parameters can be set only when creating an object or they are intended only to display information.

Possibility to changing project's parameters when the project is connected to the solver and is being calculated

General settings	✗
Comp. domain geometry	✗
Substance	✗
Physical processes	✗
Boundary conditions > Boundary condition #N	✗
Initial conditions	✗
Computational grid	✗, ✓, 1
Computational grid > Adaptation #N	✓
Simulation controls	✓
Simulation controls > Limiters	✗
Visualization layers > Layer #N	✓, ⓧ, 2
Results > Result #N	✗, ⓧ, 3

Notation signs:



- cannot be edited when connected to the solver.



- can be edited when connected to the solver (suspension of calculation is required)



- parameters are at creation of an element and cannot be changed at all, and parameters that are used only to display information.

Explanatory notes:

- Parameters of the initial grid cannot be changed when the project is connected to the solver. Adaptation parameters can be changed when the project is connected to the solver, but this requires suspending the calculation.

- 2. The Layer type and Object parameters in properties of a Layer are set at creation of the Layer and cannot be changed. Parameters from the INFO block are used only to display data.
- 3. Only the name of the Result can be changed, when the project is not connected to the solver. Other parameters of a Result (Variable, Object, Value) are set at creation of the Result created and cannot be changed.

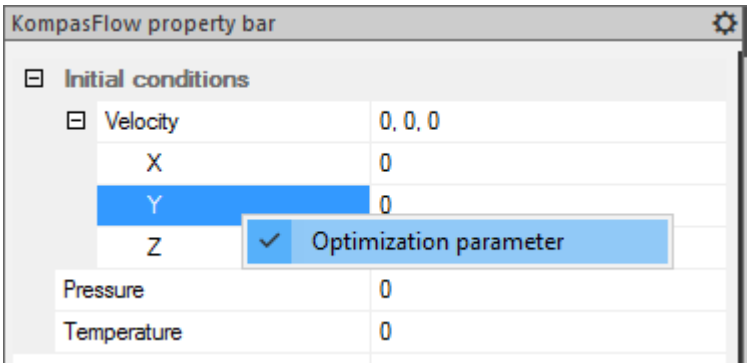
5.7 Optimization

KompasFlow includes features for connection to the Optimization IOSO-K application.

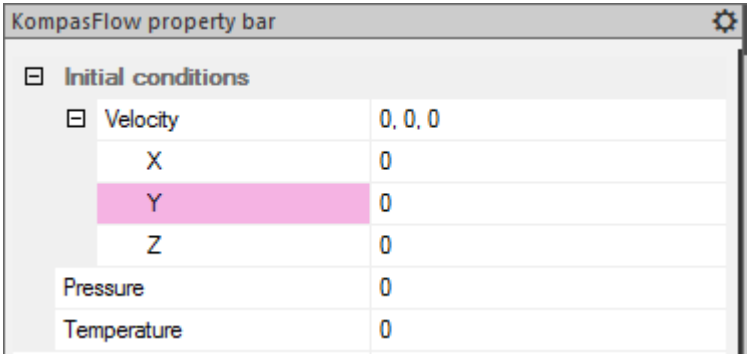
Parameters of optimization can be numerical values that are specified in properties of [Boundary conditions](#) and/or [Initial conditions](#).

Optimization parameters

Declaring an optimization parameter is done using the Optimization parameter command from the parameter's context menu:




Optimization parameters are highlighted in the properties bar with pink color:

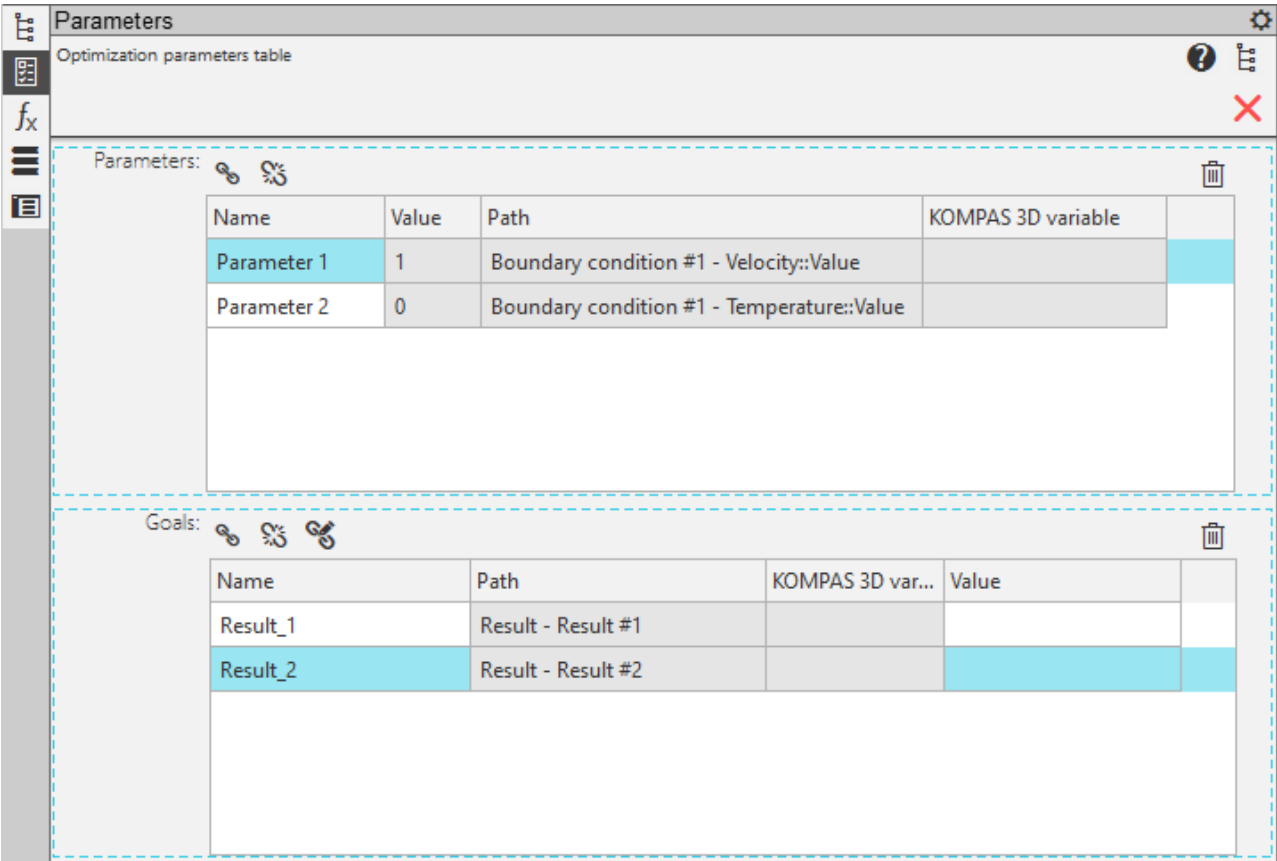


Optimization goals

One or several [Results](#) can be declared as Optimization goals. To declare a Result as an Optimization goal, open in the project tree the Result's context menu and enable the Optimization goal option.

Icons of Results, which are declared as optimization goals, are displayed as .

Optimization parameters table



To open the Optimization parameters table, click the  Optimization params. icon in the [KompasFlow toolbar](#).

The Optimization parameters table contains two tables, Parameters and Goals that contain respectively the list of parameters and the list of goals of the optimization.

Data from these tables are transferred into application Optimization IOSO-K.


The lists below represent optimization parameters.

Table "Parameters" in the Optimization parameters table

Column names:

Name	Name of the selected optimization parameter. This name can be changed and it must be unique, letters in it must be Latin, and the name must not include special symbols such as "#". By default the program prompt you to use names "Parameter 1", "Parameter 2", etc. To change a name, double click the field, enter a new name, and then press Enter on the keyboard.
Value	Value of the selected optimization parameter specified in the property bar of the appropriate element of the project tree. To enter it, double-click the field
Path	Path to the selected optimization parameter in the project tree in format Initial conditions - Parameter or Boundary condition - Physical value::Parameter .
KOMPAS 3D variable	A variable of KOMPAS-3D, which is connected to the selected optimization parameter.

Screen buttons in the Parameters table:

	Bind to a Compass	Connecting the selected optimization parameter with a KOMPAS-3D variable.
---	--------------------------	---

3D variable









	Break a connection with a Compass 3D variable	Disconnecting the selected optimization parameter from the connected KOMPAS-3D variable.
	Delete	Deleting the selected optimization parameter. The program will request you to confirm this decision (an appropriate warning will open). After deleting the selected optimization parameter its highlighting with pink color will also be removed.

Table "Goals" in the Optimization parameters table

Column names:

Name	Time of the selected optimization goal. To change the name, double-click on it, enter a new name and press the Enter key on the keyboard. <div><div>!</div><div>Only Latin characters (letters of the English alphabet) and numbers are allowed for Goal names, for example: Res1 or Goal2. Do not use special characters (for example, № or #) and spaces in Goal names, otherwise, an "Invalid variable name" error will be returned if you try to bind a Goal to a COMPAS 3D variable. A name must be unique, not the same as another Goal. To avoid problems with uniqueness of Goals and Parameters names instead of Result1, Result2 it is recommended to use underscore: Result_1, Result_2</div></div>
Path	Path to the selected optimization goal in the project tree in format "Result - Result #N".
COMPAS 3D variable	A variable of KOMPAS-3D, which is connected to the selected optimization goal.
Value	Value of the selected optimization goal. To enter it, double-click the field.

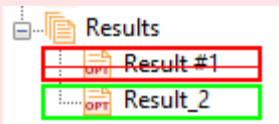
Screen buttons in Goals table:




	Bind to a COMPAS 3D variable	Connect the selected optimization goal with a <i>KOMPAS-3D</i> variable.
	Break a connection with a COMPAS 3D variable	Disconnect the selected optimization goal from the connected <i>KOMPAS-3D</i> variable.
	Update COMPAS 3D variable values	Update values of <i>KOMPAS-3D</i> variables. This can be done only when the program is connected to the solver (otherwise a warning " No connection to solver " will be output).
	Delete	Deleting the selected optimization goal. The program will request you to confirm this decision (an appropriate warning will open). After deleting the selected optimization goal, icon of the appropriate element Result #N in the project tree will change  again instead of  .

!

When a new Goal is created, its name is copied from the name of the corresponding **Result** in project tree ("Result #1", "Result #2", etc.). Such spelling violates the program requirement for connection with a COMPAS 3D variable, for correct operation do the following (choose one method):

- When creating a new Result, set a name that matches the requirements for the Goal name (Latin letters and numbers, no spaces and symbols)
- After creating the Goal, go to the Optimization parameters table and rename the created Goal (delete the space and the symbol "#")



Goals:   


Name	Path	KOMPAS 3D variable	Value	
Result #1	Result - Result #1			
Result2	Result - Result #2			
Result_3	Result - Result #3	kfout_Result_3	0	

If the name of a Goal meets the specified requirements, but the link is still cannot be created, the name is most likely not unique. Specify a different name and try to create a bind again.



5.8 Creating reports



The program can generate auto reports that include general information on the project. Such auto reports can be used as an initial approach to creating a more informative and detailed report.

An auto report can be generated in various formats (.odt, .pdf), in various styles (Standard, GOST), and it will include specified groups of information from the project and a screenshot.

To generate a report apply the command  Create report from the Additional commands [toolbar](#).

A form will open containing parameters of the auto report.

File path:

Report format:

Reporting style:

^ Properties for the report

General settings: ☐

Substance: ☐


Physical processes: ☐


Boundary conditions: ☐

Initial conditions: ☐

Computational grid: ☐



Simulation controls: ☐






This form contains the following field and settings:

File path	Path to the file, which will contain the generated report. This is a read-only field.
Report format	Format of the report file. Possible options are: <ul style="list-style-type: none">• .odt• .pdf

Reporting style	<div>Style of the report. Possible options are:</div> <ul style="list-style-type: none">• Standard• GOST
Properties for the report	<div>These are settings that specify groups of information, which will be included into the report (the report will also include a screenshot). Screen toggles, which set if the group of information would be included into the report, locate next to names of the groups:</div> <ul style="list-style-type: none">•  – this information will be included into the report.•  – this information will not be included into the report. <div>To toggle these settings, click them with the mouse.</div> <div>The groups of information, which can be either included or not included into the report, are:</div> <ul style="list-style-type: none">• General settings• Substance• Physical processes• Boundary conditions• Initial conditions• Computational grid• Simulation controls

When you specify all the desired parameters, click the icon . The program will generate a report according to the specified parameters and place it into the folder specified in the field File path.

Example of an auto report

Project Name: mlwark_rus
KompasFlow Version: 23.0.1
Date: 09.05.2024 16:31:31

1 General settings

Reference pressure	101000
Reference temperature	273
Gravity vector	(0, 0, 0)
Hydrostatics	No
Tolerance of geom. calc.	1e-8
External task	No

2 Substance

State of aggregation	Gas
Ideal gas law	Yes
Molar mass	0.029
Viscosity	2e-5
Thermal conductivity	0.022
Specific heat	1000

3 Physical processes

Equation of motion	Yes
Turbulence	No
Heat transfer equation	Yes
Radiation	No

4 Boundary conditions

4.1 Граничное условие

Name	Граничное условие
Color	
Type BC	Wall
Roughness	0

4.1.1 Variables

4.1.1.1 Temperature

Method	Heat flux
Value	0

4.2 Boundary condition #1

Name	Boundary condition #1
Color	
Type BC	Inlet/Outlet

4.2.1 Variables

4.2.1.1 Velocity

Method	Normal mass velocity
--------	----------------------

Value

1

4.2.1.2 Temperature

Value

0

5 Initial conditions

Velocity	(0, 0, 0)
Pressure	0
Temperature	0

6 Computational grid

Adaptation to solution	No
------------------------	----

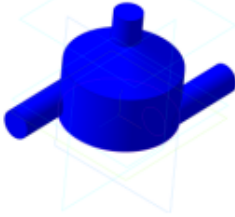
6.1 Initial grid

n1	10
n11	10
n2	10

7 Simulation controls

Method	Via CFL number
CFL	10
Max. time step	0.1
Simulated time	10
Save frequency	50

8 Screenshot



Picture 1 - Screenshot

6 Problem solving

[Possible problems](#)

[Error messages and warnings](#)





[Technical support](#)

6.1 Possible problems

<i>Problem</i>	<i>Recommendations</i>
Incorrect work with one document in multiple windows	It is recommended not to operate with one document in several windows.
Modification of the model's geometry sometimes is not tracked in KompasFlow	After modifying the geometry, it is recommended to carry out its manual check.
Sometimes modifications of the geometry (scaling, moving) can change parameters of boundary conditions.	After modifying the geometry, check and, if necessary, set parameters of boundary conditions again.

If an error message or warning appears, see section [Error messages and warnings](#).

6.2 Error messages and warnings

Error message or warning	Description
Could not find a suitable body to create a CFD-task.	<p>This message is displayed when the program was not able to find a topologically sewn body in the geometrical model.</p> <p>If the body can be formed from already existing individual surfaces sewn together, apply the  Knit surfaces command from the  Wireframe and surfaces tool bar of KOMPAS-3D.</p>
The previous calculation data will be lost! Continue?	<p>This warning is displayed before starting a calculation from scratch by the  Start solve command. If the answer is No or Cancel, you will have a possibility to load the saved calculation data using the  Open the solution command.</p>
The task has been saved for another version of geometry	<p>When the program starts, it compares the stored and current geometry. If a discrepancy is found, a notification appears. The system automatically tracks the version of your project's geometry. For each face, geometric data and a change-tracking task are stored in the .m3d file. Any new faces created are assigned the Wall boundary condition by default. If a Wall condition is not defined in the project, new faces receive the first available boundary condition instead. When a calculation begins, the current geometry version is sent to the solver and archived within the project files.</p>
Geometry of the model has changed, disconnecting from solver	A notification appears if changes have been made to the geometry with the solver enabled. This disconnects from the solver.
The calculation data was created for another version of the geometry, the connection is prohibited!	A notification appears if the geometry versions do not match when trying to upload calculation data. The comparison takes place between the current version of the geometry and the one from which the calculation was performed.

6.3 Technical support

If necessary, contact KompasFlow technical support.

Requests and suggestions for KompasFlow are appreciated; send them to KompasFlow technical support.