FlowVision

Tutorial: Examples of typical tasks

Version 3.13.01

Contents

1 Work with this tutorial

This tutorial is intended to form your skills of making typical simulations using the *FlowVision* software.; the tutorial contains the following parts:

1. the first part ("Detailed description of a simplest model") contains a detailed description of all steps of specifying a project, of carrying out the computation and analysis of the simulation's results. As a simplest example we selected simulation of a laminar flow in a tube.
2. the second part ("Physical processes") illustrates use of different basic physical processes.
3. the third part ("Advanced modules") illustrates use of additional features.

The most efficient way to learn *FlowVision* is doing successive exercises (examples) from the tutorial. The tutorial, using these successive examples, familiarizes you with all main and additional features of *FlowVision*.

It is highly recommended to start your learning from the section "*Quick start"* from the *"User's guide"* document; then do the successive exercises from the tutorial's sections *"Detailed description of a simplest model"* and *"Physical processes".*

We also recommend you to do all exercises from the section "*Advanced modules*" or at least those, which are most close to problems from your field.

For each example use the following materials:

- description of the problem in the tutorial, see appropriate sections of the tutorial
- files with the geometry, which are included in the distribution package of *FlowVision* (they locates in the folder **`Tutorial\Samples\Geom`** if standard location settings have been used).
- files of client part of the project, which are also included in the distribution package of *FlowVision* (they locates in appropriate subfolder in the folder **`Tutorial\Samples\EnuProjects`** if standard location settings have been used).

In exercise projects delivered along with the program sequence of boundary conditions in the project tree and their colors might differ from those given in the tutorial.

Before you start the exercises, make sure that:

- client and server *FlowVision* modules have been installed
- **License Manager** is installed
- the license is obtained and registered
- all settings of **License Manager** and **Solver-Agent** are tuned
- **Solver-Agent** and **License Manager** are running
- directories for server parts and client parts of projects have been created
- the **Solver-Agent**'s user have been created

(see details in the *"User's guide"* document).

Date and time of the document's creation: 12/15/2022, 3:41 PM.

2 Notation

Notation	Quantity	Name in *FlowVision*	Dimension
a	Sonic speed		$m\ s^{-1}$
b	Initial or entrance flow turbulization	**Pulsation**	
CFL	Courant number	**CFL**	
C_p	Specific heat capacity		$m^2\ s^{-2}\ K^{-1}$
D	Diameter of a tube		m
f	A scalar variable		
k	Turbulent energy	**TurbEnergy**	$m^2\ s^{-2}$
L	Length of a tube		m
l	Initial or entrance turbulence length scale	**Turbulent length scale, m**	m
$M = U/a$	The Mach number	**MachNumber**	
m	Mass of a body		kg
p	Relative static pressure	**Pressure**	$N\ m^{-2}$
Pr	Molecular Prandtl number		
Pr_t	Turbulent Prandtl number	**Prandtl**	
$Re = \frac{UD\rho}{\mu}$	Reynolds number		
Sc_t	Turbulent Schmidt number	**Schmidt**	
T	Temperature	**Temperature**	K
U	Characteristic velocity		$m\ s^{-1}$
u_τ	Friction velocity		$m\ s^{-1}$
V_{inl}	Entrance flow velocity		$m\ s^{-1}$
V_{ini}	Initial flow velocity		$m\ s^{-1}$
y	Distance to the nearest wall	**DistanceToWall**	m
$y^+ = \frac{u_\tau y}{\nu}$	Dimensionless distance to the nearest wall	**Y_plus**	
ε	Dissipation rate of turbulent energy	**TurbDissipation**	$m^2\ s^{-3}$
λ	Molecular heat conductivity		$kg\ m\ s^{-3}\ K^{-1}$
μ	Molecular dynamic viscosity		$kg\ m^{-1} s^{-1}$
μ_t	Turbulent dynamic viscosity	**TurbViscosity**	$kg\ m^{-1} s^{-1}$
$\nu = \mu/\rho$	Molecular kinematic viscosity		$m^2\ s^{-1}$
$\nu_t = \mu_t/\rho$	Turbulent kinematic viscosity		$m^2\ s^{-1}$
ρ	Density	**Density**	$kg\ m^{-3}$
$\tau = \rho u_\tau^2$	Viscous stress at a wall	**Shear Stress**	$N\ m^{-2}$

Notation	Quantity	Name in *FlowVision*	Dimension
τ	Time step		s
$\omega = \dfrac{\varepsilon}{\beta^{*} k}$	Specific dissipation rate of turbulent energy	**TurbDissipation**	s^{-1}

3 Detailed description of a simplest model

This chapter describes in detail all the steps in specifying a *FlowVision* project, performing calculations, and analyzing results. The control settings are defined in **Preprocessor**, the analysis tools are presented in **Postprocessor**.

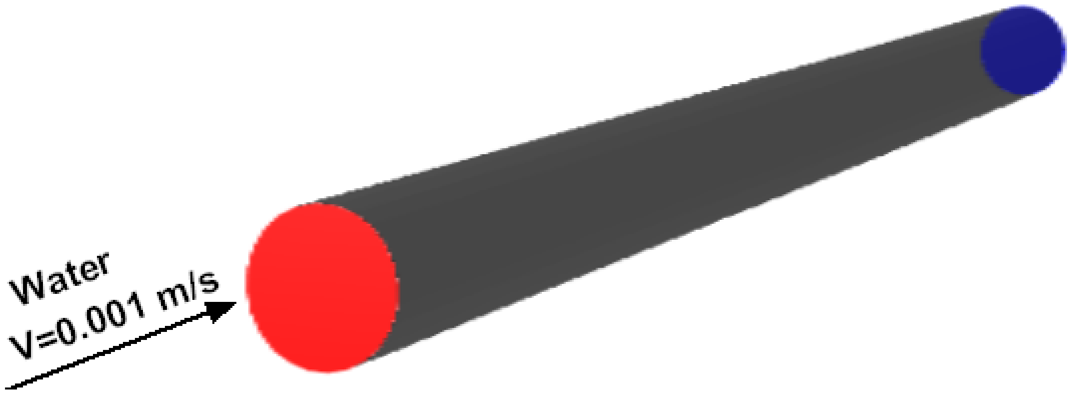
It is recommended to start your work with this example.

3.1 Laminar flow in a tube

This exercise teaches you how to create a project, start computations, and analyze results.

A laminar flow of a viscous fluid in a cylindrical tube is considered.

The laminar flow is characterized by the Reynolds number Re, when it is lower then 10^3.



Dimensions:				
	Length of the tube	L	= 2	[m]
	Diameter of the tube	D	= 0.1	[m]
Inflow parameters				
	Velocity on the inlet:	V_{inl}	= 0.001	$[m \cdot s^{-1}]$
Substance properties:				
	Density	ρ	= 1000	$[kg\ m^{-3}]$
	Viscosity	μ	= 0.001	$[kg\ m^{-1} \cdot s^{-1}]$
Reynolds number:		$\mathrm{Re} = \frac{V_{inl} D \rho}{\mu} = \frac{0.001 \cdot 0.1 \cdot 1000}{0.001} = 10^2$		
Geometry:		**`Tube.wrl`**		
Project:		**`Lam_tube`**		

3.1.1 Computational domain

The geometrical model of the computational domain is created in one of CAD systems and imported into *FlowVision*.

The geometric model of the computational domain must satisfy the following requirements:
1. volumes that form the geometric model must be closed
2. volumes are nested in one another and do not intersect

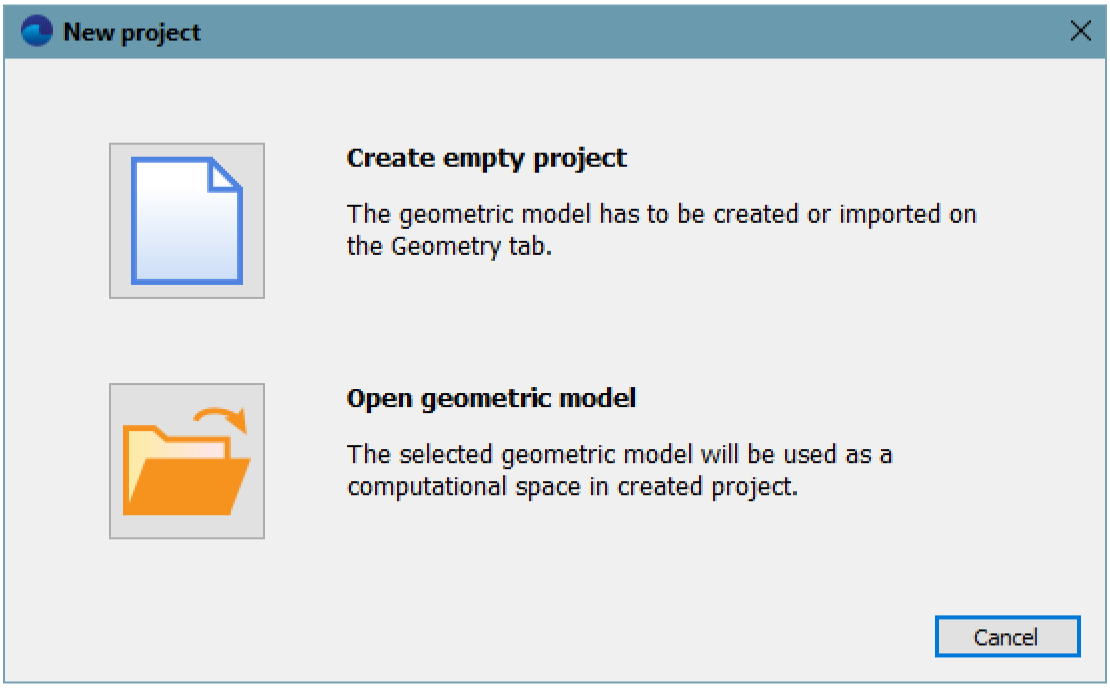
The geometric model is transferred to *FlowVision* using one by one of the following standard formats:
1. surface mesh: *VRML*, *STL*, *MESH* (internal format)
2. volume mesh, based on which in *FlowVision* a corresponding surface mesh is built: *ANSYS*, *NASTRAN*, *ABAQUS*

Fully prepared and painted the geometry of the computational domain is stored in the file **`Tube.wrl`**.

3.1.2 Creating a project

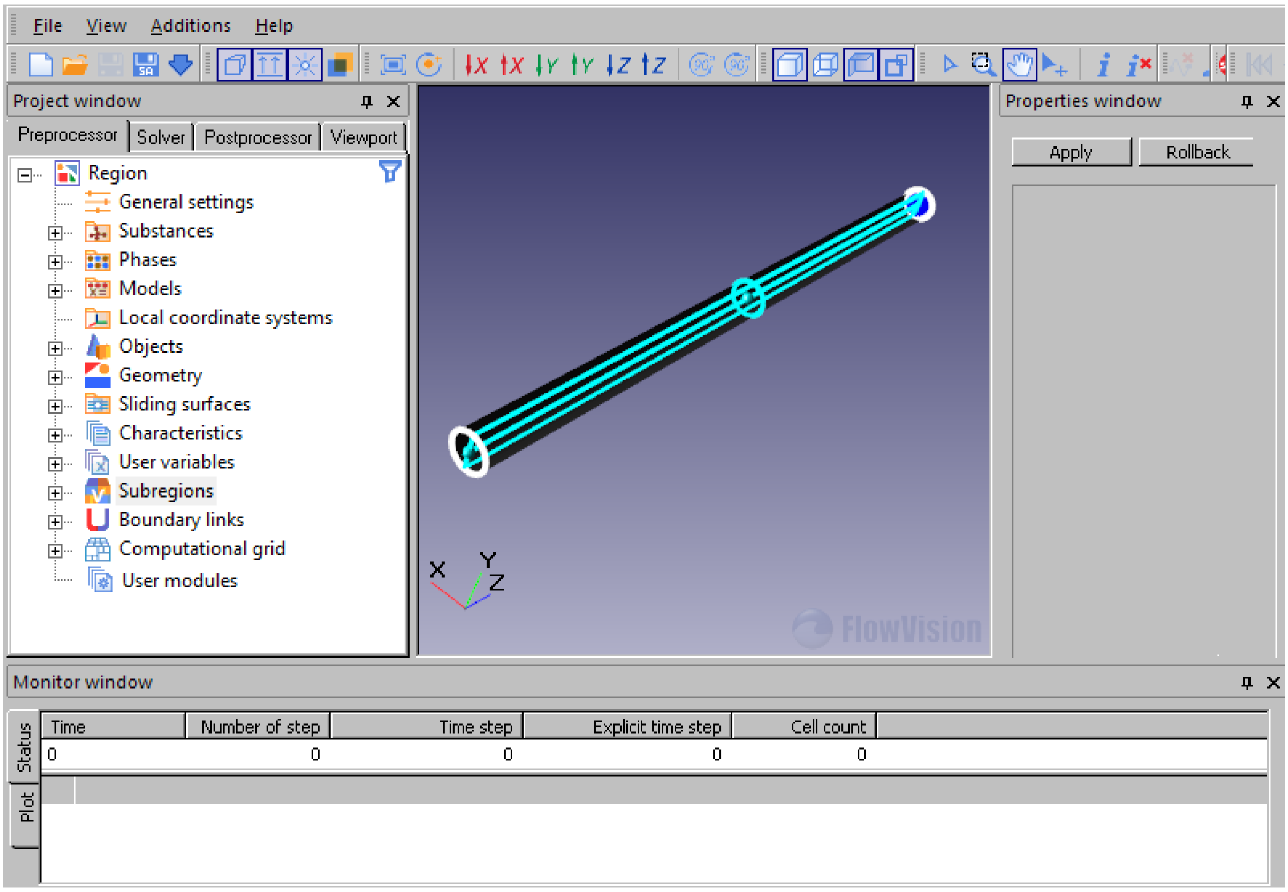
To load the geometry model into *FlowVision*, apply the **File > Create** command from the main menu.

The **New project** dialog box will open:



In this dialog box click the  (**Open geometric model**) button. A standard operating system's dialog box for selecting a file will open where you have to select the file, which contains the geometry model of the computational domain. Select the `Tube.wrl` file there.

After that the geometric model will appear in the **View** window of **Pre-Postprocessor**, and appropriate **SubRegions** will appear in the **Project** window, in the **Preprocessor** tab.

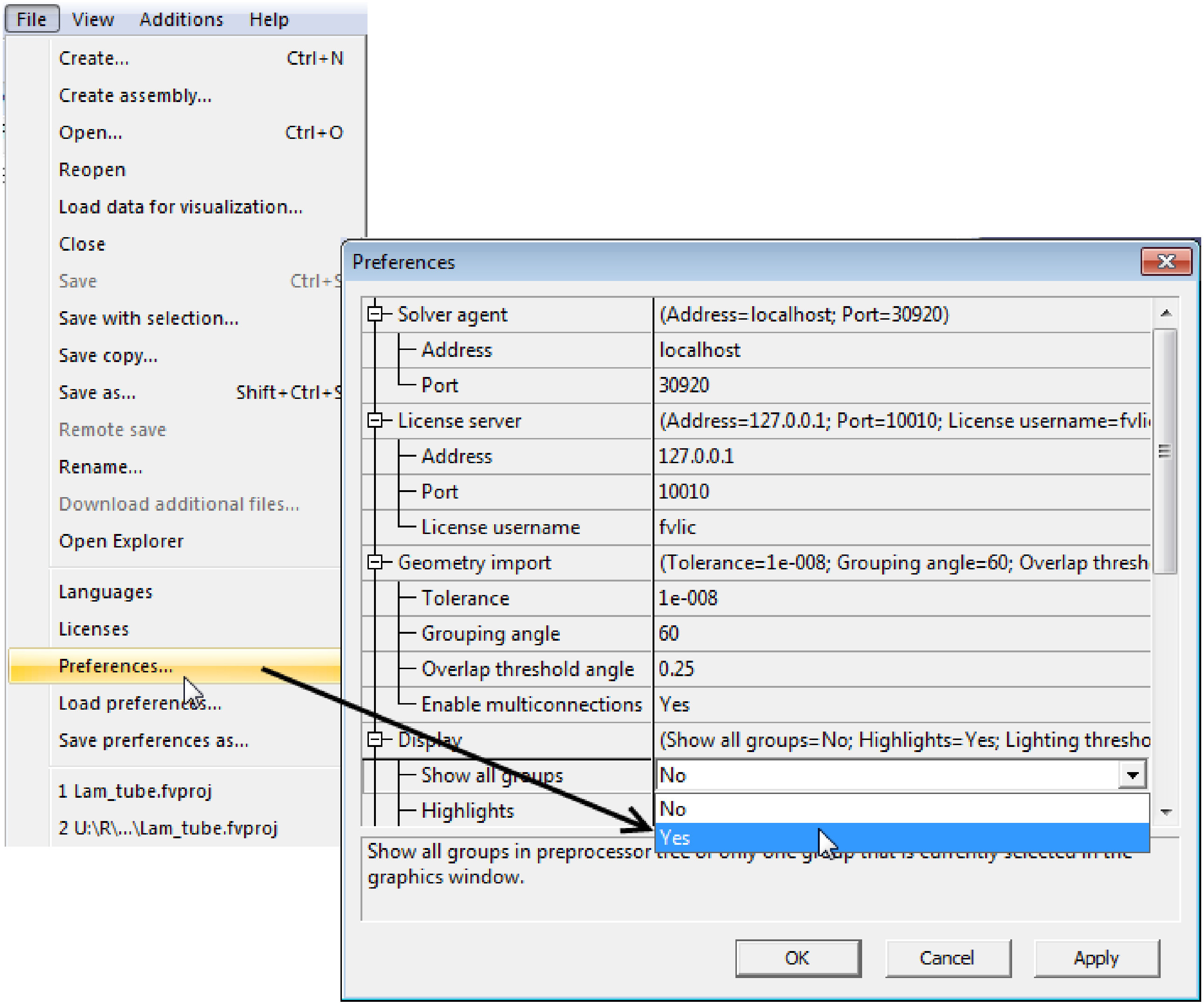


The program loads the geometry of the surface as a set of triangles. After loading the geometry, the triangles are automatically merged into geometric group.

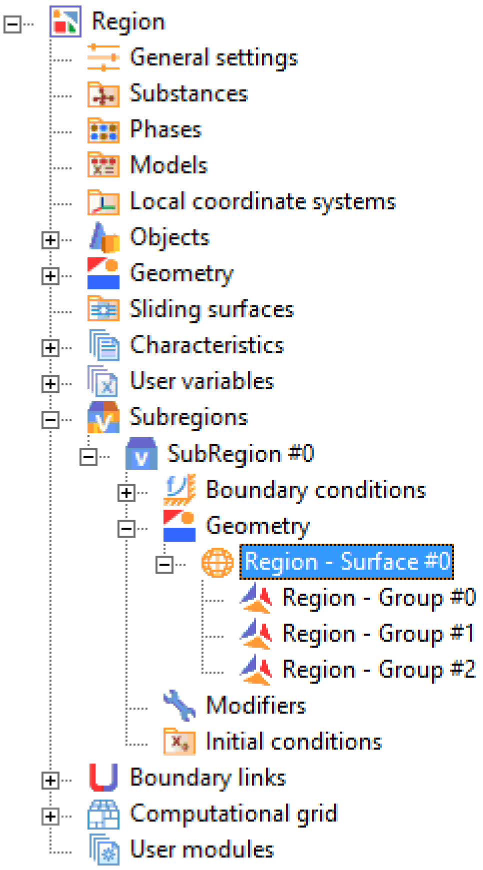
In order to show **Groups** in the project tree, specify using the menu command **File > Preferences**:

Display

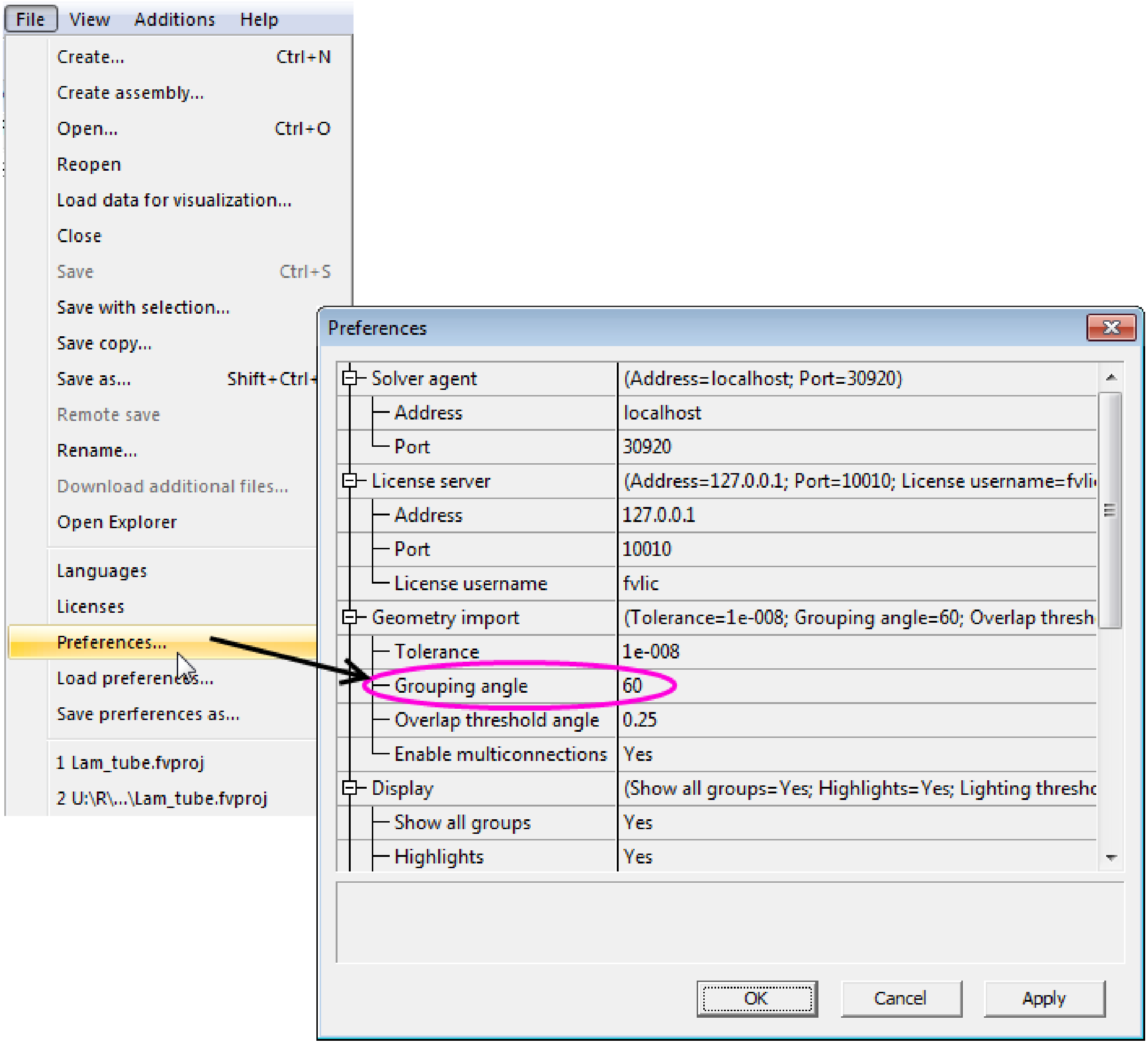
Show all groups = Yes



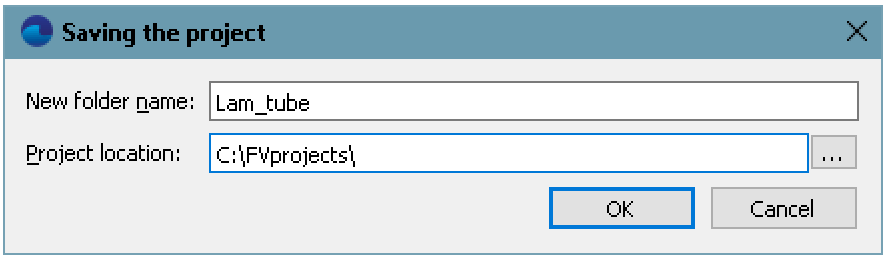
List of **Group** is displayed in a tree in the **Preprocessor** tab in the folder **Region > Subregions > SubRegion#0 > Geometry > Region - Surface #0**.



The number and boundaries of the groups depend on the geometry format and settings of **Pre-Postprocessor**. If geometry format supports color information (eg `.wrl`), then the triangles of the same color will be merged into a single group. If the format does not support color information (eg, STL), then the triangles are arranged in groups according to the magnitude of the **Grouping angle**. If the angle between the triangles is less than the **Grouping angle**, they are combined into one group. This angle is specified by the parameter **File > Preferences > Geometry import > Grouping angle**.



Save the project on the disk using the **File > Save** command from the main menu or press **Ctrl+S** on the keyboard. The **Saving the project** window where you have to specify the project's name in the **New folder name** field and location of the folder with the project's files in the **Project location** field; then click **OK**:



The folder with the project's files will be created.

3.1.3 Defining a physical model

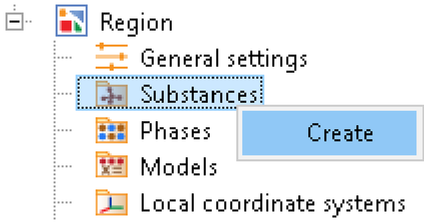
To specify a physical model, it is necessary to define:

- Substance
- Phase
- Model

3.1.3.1 Substance

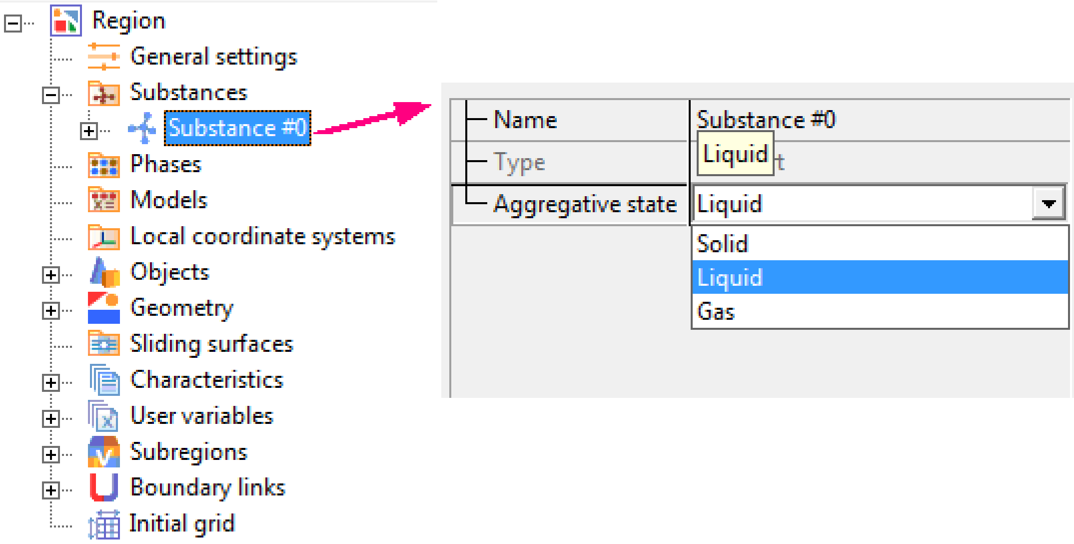
In this example, a water flow is simulated, so we have to define one substance - water.
In order to specify a new substance, perform the following steps:

- In the **Preprocessor** tab in the context menu of the **Substances** folder, select **Create**.



This will create the **Substance #0** folder with appropriate parameters.

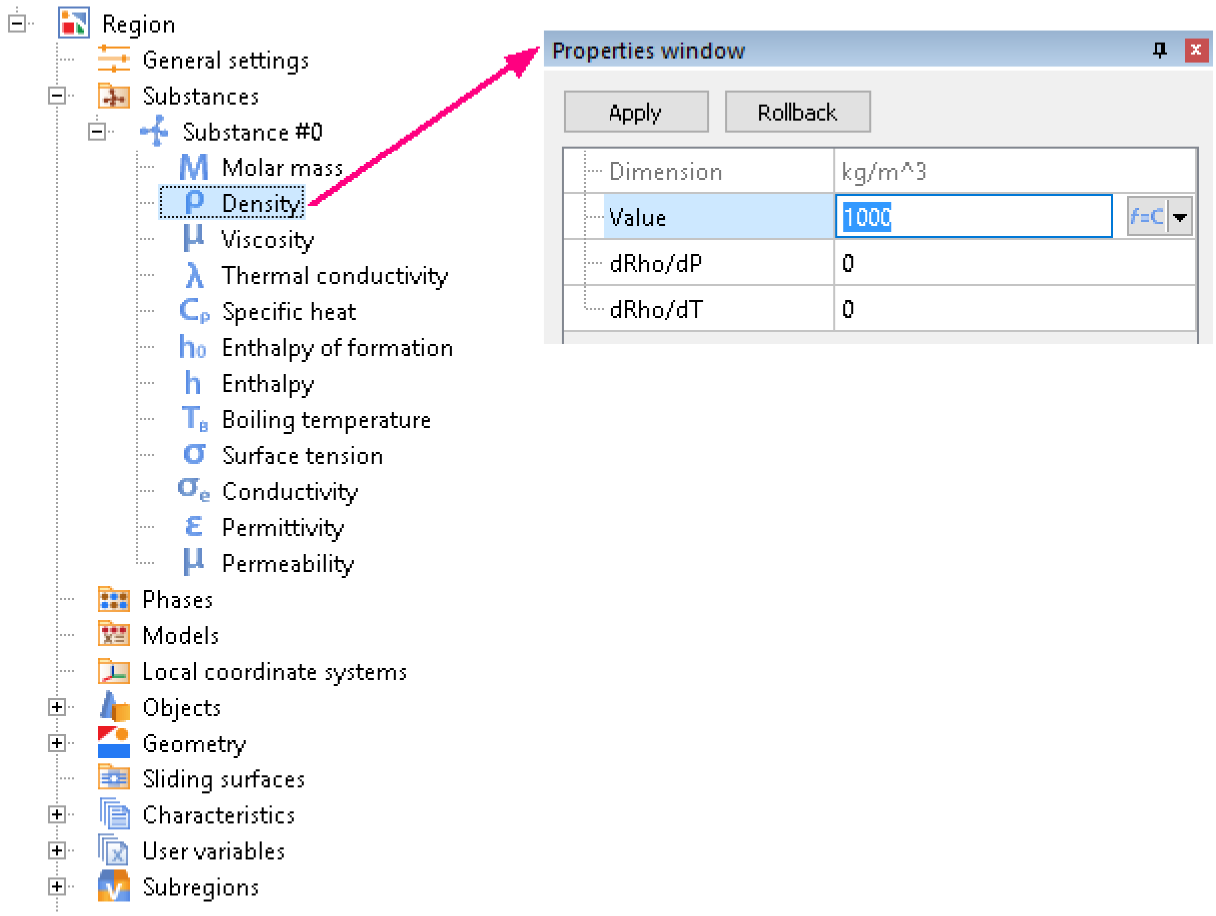
- In the **Properties** window of **Substance #0** in the **Aggregative state** field select **Liquid**:



- In the folder **Substance #0** in the list of the physical properties of the substance you have to specify:

Molar mass		
Value	**=0.018**	$[kg\ mole^{-1}]$
Density		
Value	**= 1000**	$[kg\ m^{-3}]$
Viscosity		
Value	**= 0.001**	$[kg\ m^{-1}\ s^{-1}]$
Specific heat		
Value	**= 4217**	$[J\ kg^{-1}\ K^{-1}]$

We do not have to specify other parameters, so you can leave them with their default values.



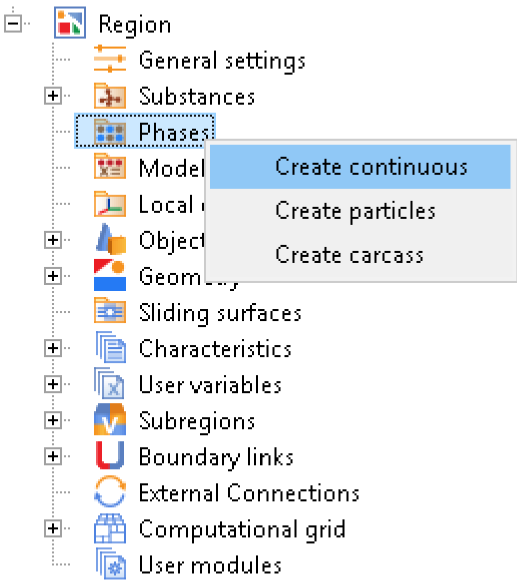
Note:

The aggregative state determines the list of physical properties, the sonic speed, and the equation of state.

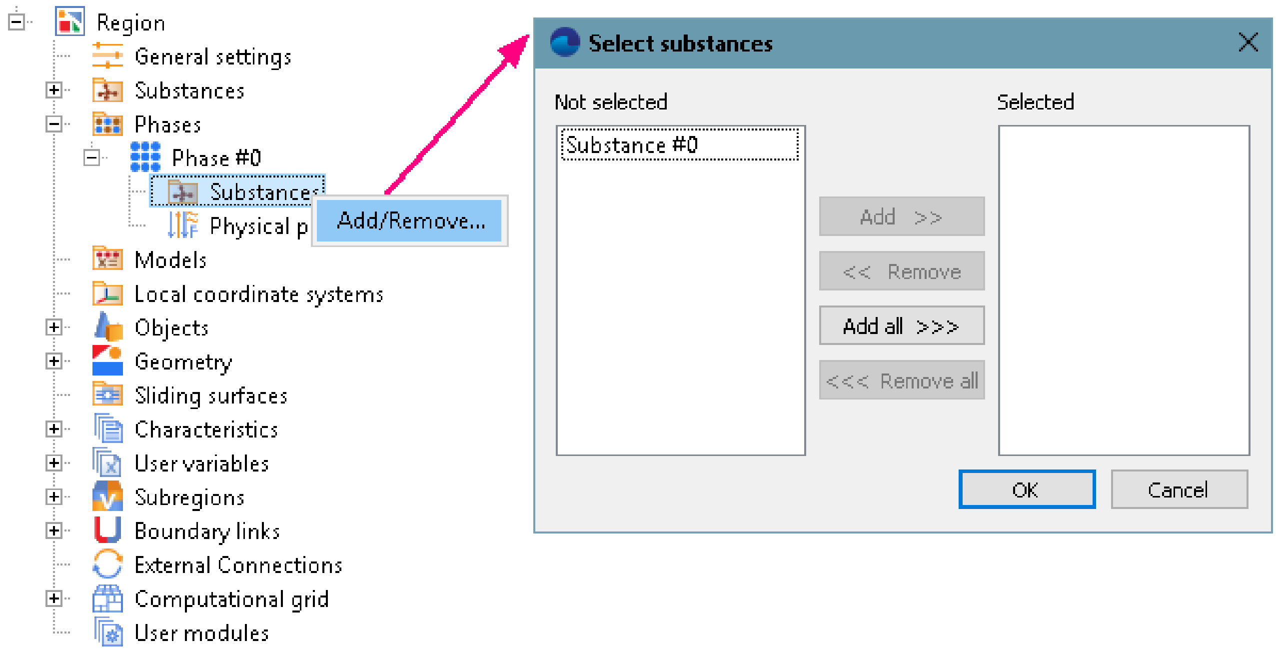
3.1.3.2 Phase

To define the **Phase**, do the following:

- From the context menu of the folder **Phases** select **Create continuous**.

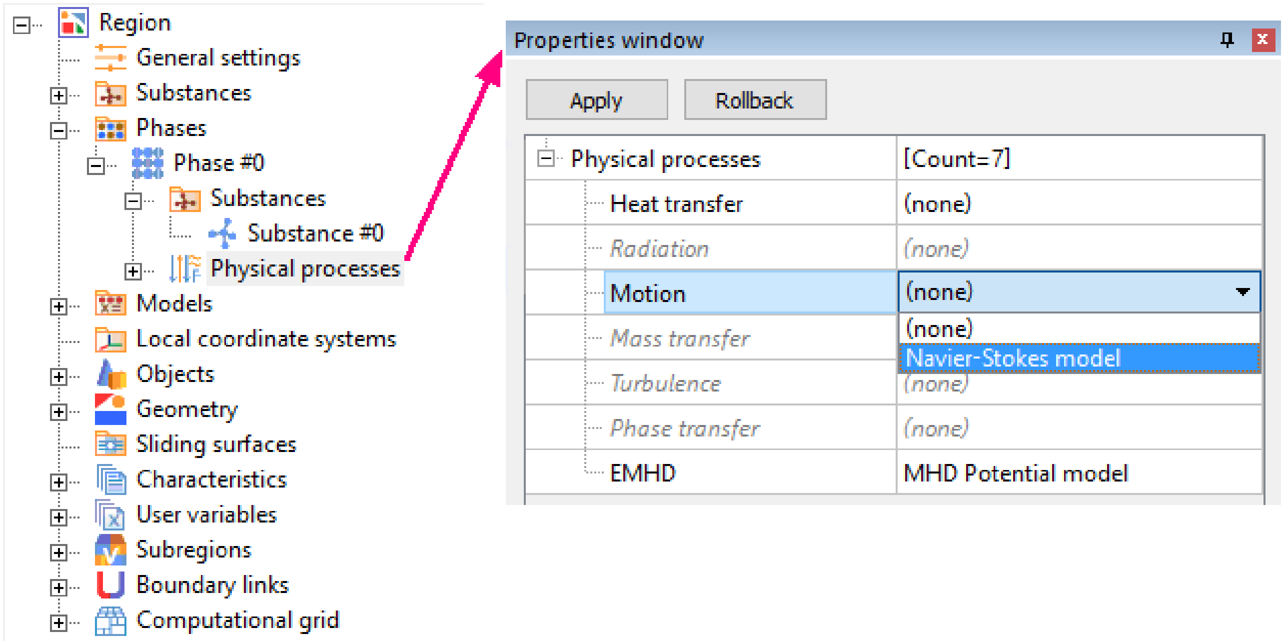


- In **Phase #0** in the folder **Substances** load a previously created substance:
  - From the context menu of the **Substances** folder select **Add/Remove**
  - Select **Substance #0** from the list and add it to the phase:



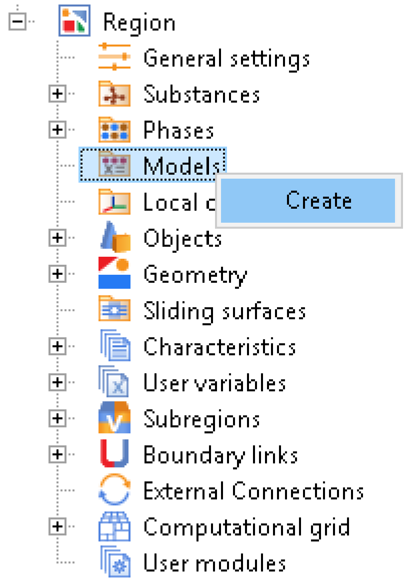
- In the folder **Physical processes** in the **Properties** window select the simulated processes. In this task the following physical processes are required:

Motion **Navier-Stokes model**

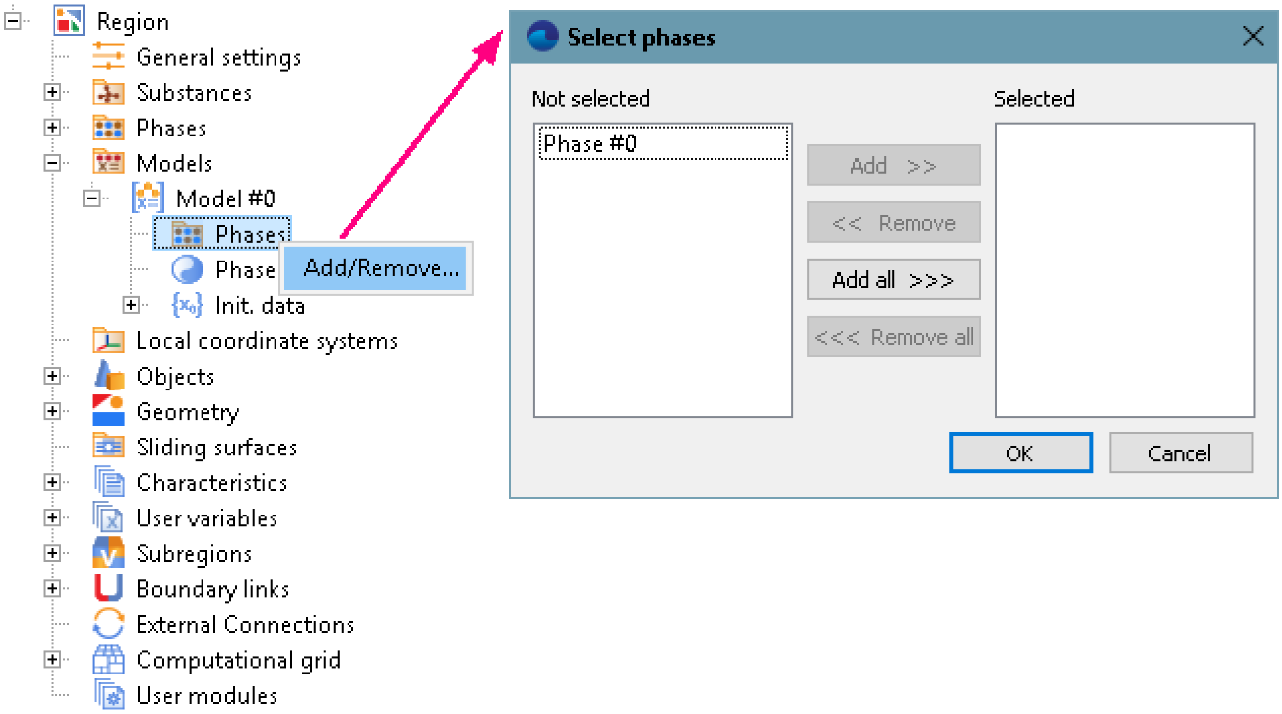


3.1.3.3 Model

In order to specify the **Model** select **Create** from the context menu of the **Models** folder.



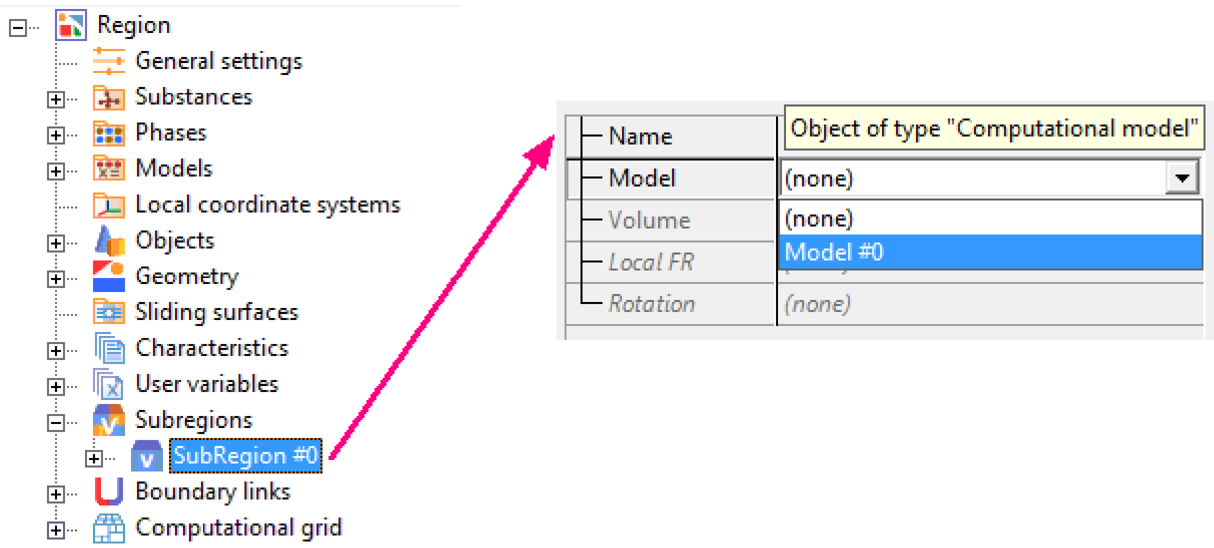
This will create the **Model #0** folder and its subfolders **Phases** and **Phase interaction**. In this exercise we need only one phase. In order to add it to the **Model** you need in the context menu select **Add/Remove** and select **Phase #0** from the list.



3.1.4 Defining boundary conditions

Before setting the boundary conditions, it is necessary to specify the model for computational subdomain. Specify in the **Properties** window of **SubRegion #0** the following parameters:

Model = **Model #0**

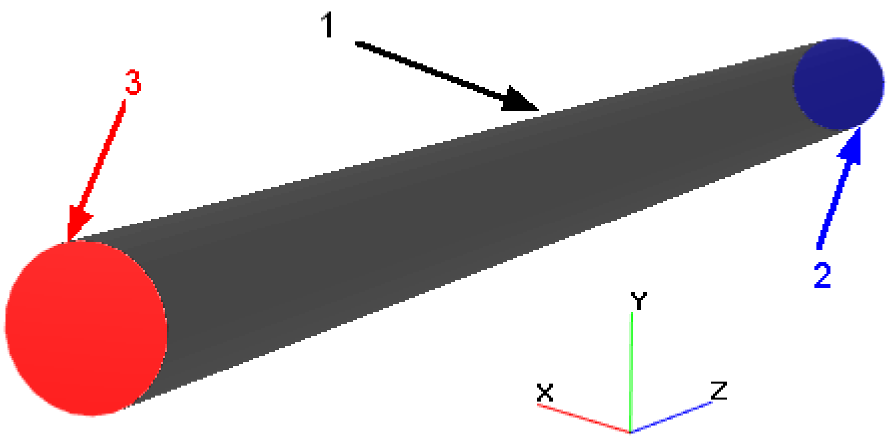


Boundary conditions are specified for every calculated variable.

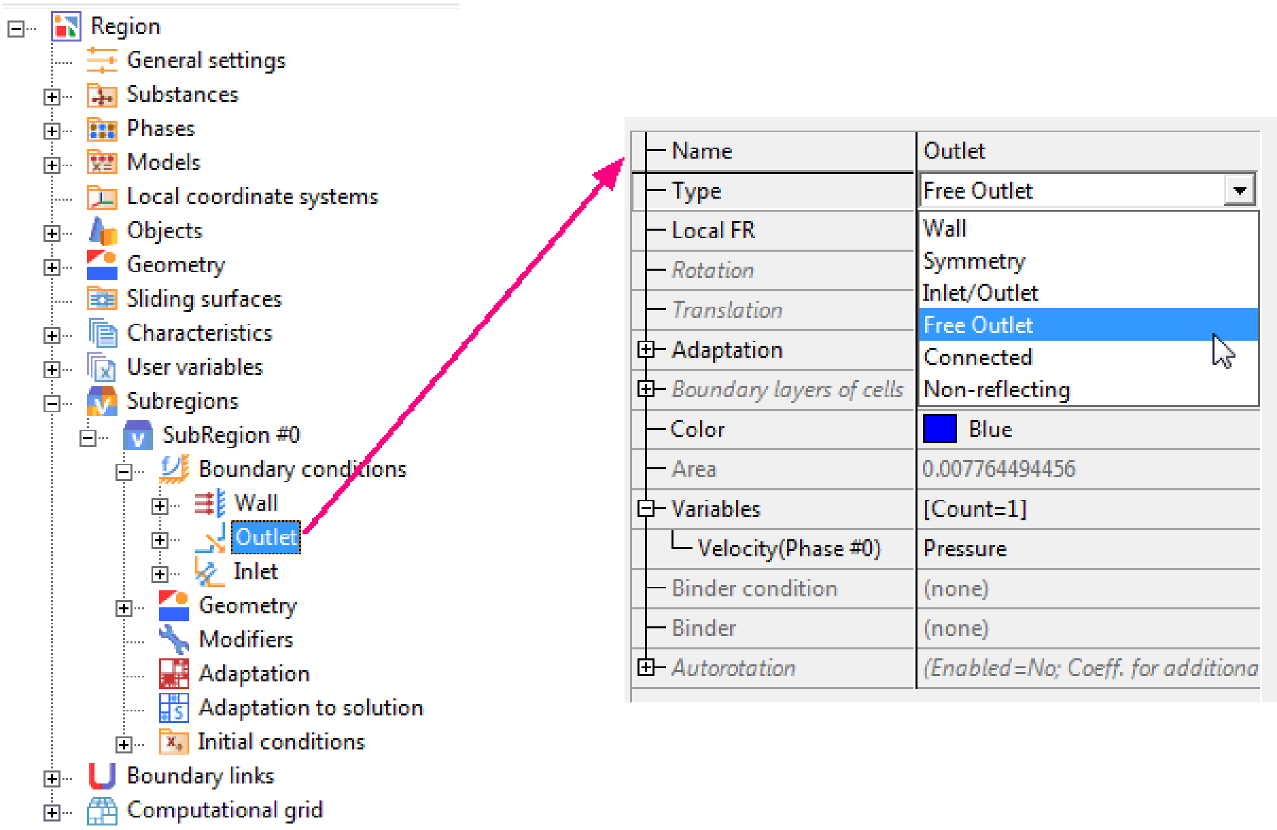
In general, the procedure for setting the boundary conditions includes the following steps:

1. Creation boundary conditions
2. Assigning the boundary conditions
3. Setting parameters of the boundary conditions

The initial geometry for this problem was painted during its creation and stored in the `.wrl` format, which supports color information. Therefore, when loading the geometry, the program automatically created and specify the boundary conditions on the surfaces of different colors. By default, all the boundary conditions are of type **Wall**.



Changing the properties of boundary conditions is done in the **Properties** windows of elements of the folder Boundary conditions in the **Preprocessor** tab of the project tree: **Subregions > SubRegion #N > Boundary conditions > B. Cond. #N**.



Specify the following parameters in the **Properties** windows of the boundary conditions:

Boundary 1

Name **= Wall**

Type **= Wall**

Variables

Velocity (Phase #0) = **No slip**

Boundary 2

Name **= Outlet**

Type **= Free Outlet**

Variables

Velocity (Phase #0) = Pressure

Value = 0

Boundary 3

Name **= Inlet**

Type **= Inlet/Outlet**

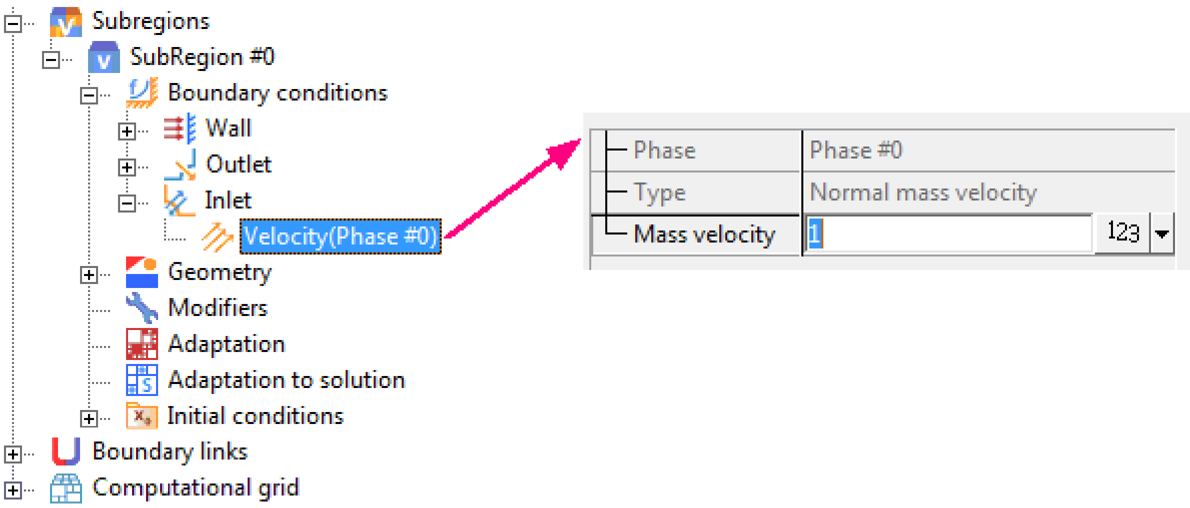
Variables

Velocity (Phase #0) **= Normal mass velocity**

Specify the numerical value of the **Mass velocity**:

Velocity (Phase #0)

Mass velocity = **1** [kg m^{-2} s^{-1}]



3.1.5 Defining initial conditions

Initial conditions are intended to define the values of the variables (**Init. data**) at the initial time in a certain part of the computational domain (in/on some **Object**). Specifying **Initial conditions** in most cases does not affect the final result of the calculations, but allows faster convergence of the solution and prevents some disturbances occurring in the process of computation.

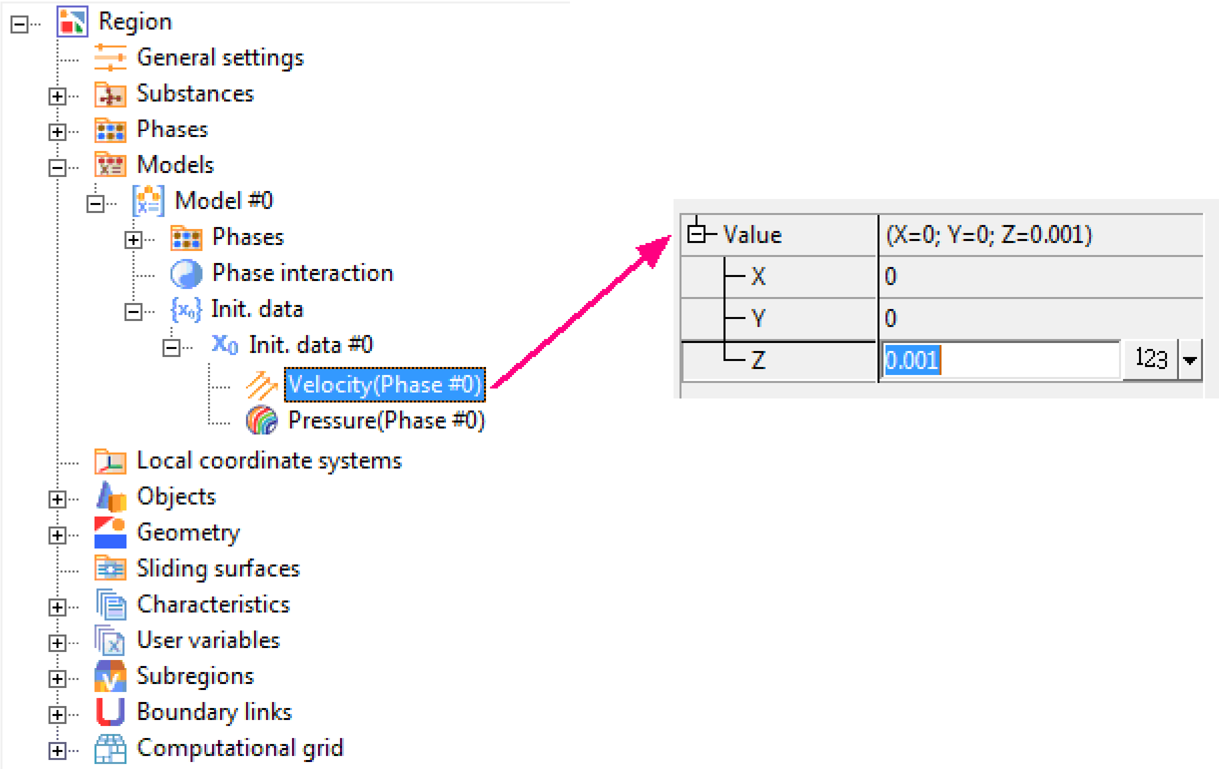
The process of creating **Initial conditions** consists of three stages:

- specifying **Initial data**
- specifying an **Object** in/on which the **Initial data** would be applied
- assigning a correspondence between the **Object** and the **Initial data**

Initial condition #0 are always presented in the computational domain, they are the default correspondence between **Init. data #0** and **Computational space**. Values of all variables in the **Init. data #0** are equal to 0. Therefore, in order to specify the initial conditions corresponding to the undisturbed flow in the whole space, it is enough to specify **Init. data #0** (in the **Model #0**) with some velocity along the Z axis.

- In the folder **Models > Model #0 > Init. data > Init. data #0**, in the element **Velocity(Phase #0)**, define the initial velocity along the axis Z:
  Value
  Z **0.001**

.



3.1.6 Generation of initial computational grid

The next necessary step is defining of the computational grid.

Computational grid used in *FlowVision* is:

1. rectangular
2. adaptive local refined [1)]
3. with a subgrid resolution of the geometry [2)]

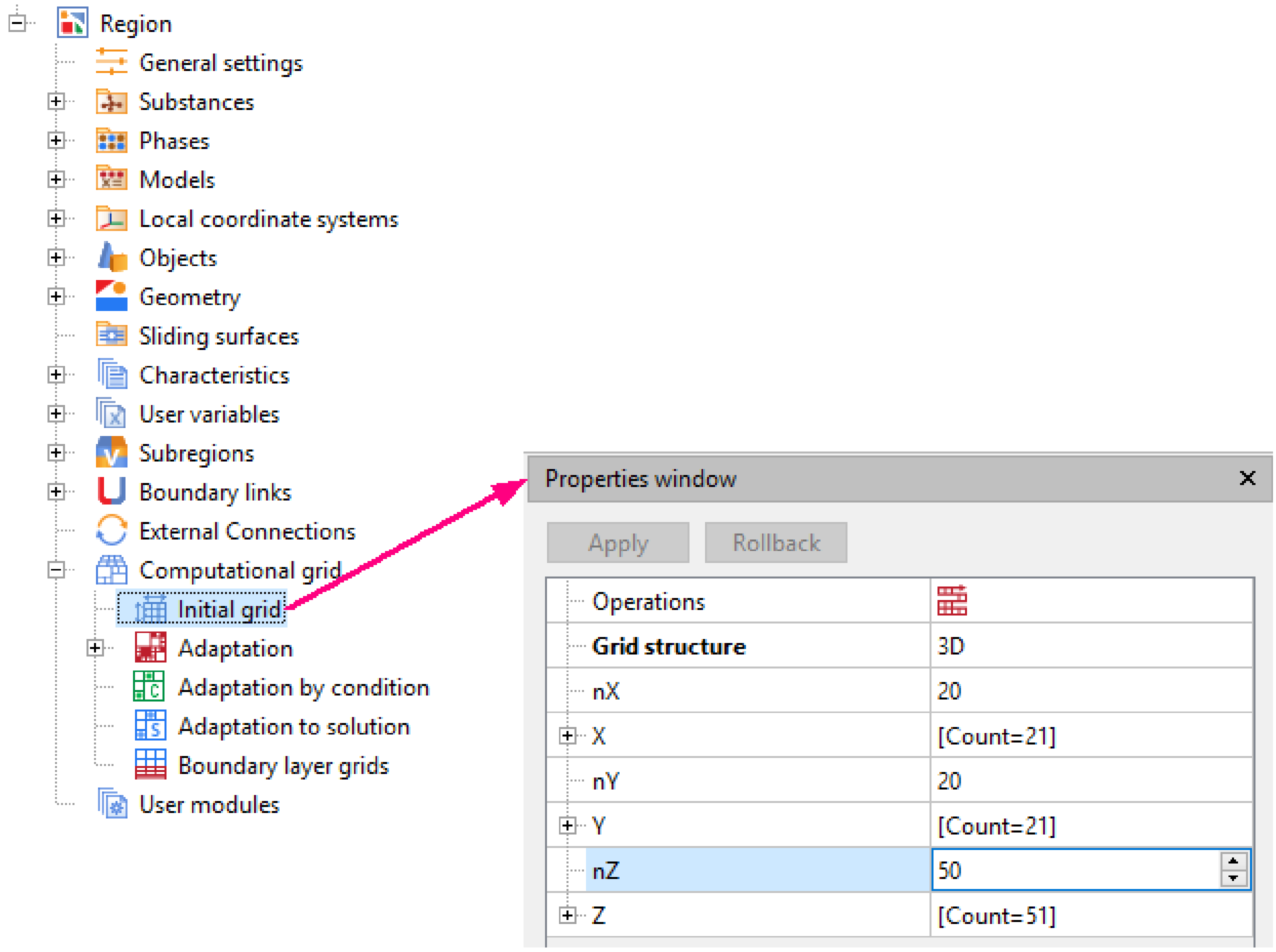
Specifying the **computational grid** in *FlowVision* is divided into specifying of the **Initial grid** (the entry level grid) and the grid adaptation (on a surface and/or in a volume).

In this example, we need to specify a uniform computational grid 20x20x50. To do this, in the project tree in the **Preprocessor** tab specify the properties of the element **Computational grid > Initial grid**:

nX = 20

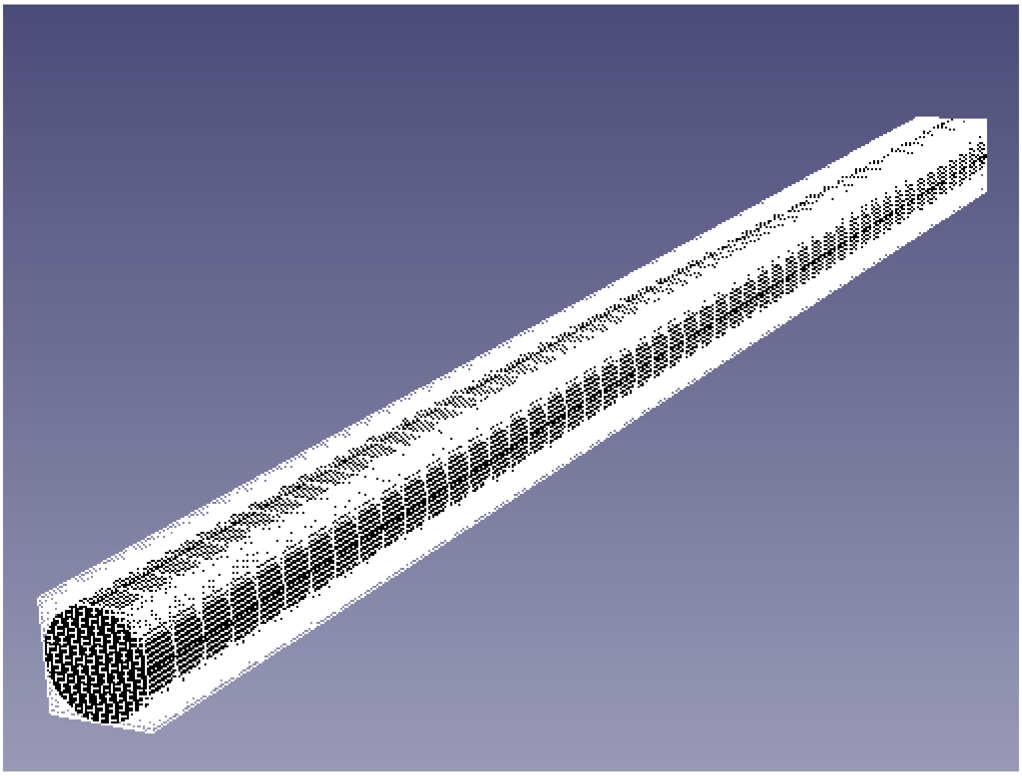
nY = 20

nZ = 50



In the **Properties** window of the **Initial grid** click **Apply**.

After you have specified the initial grid, it is displayed in the **View** window:

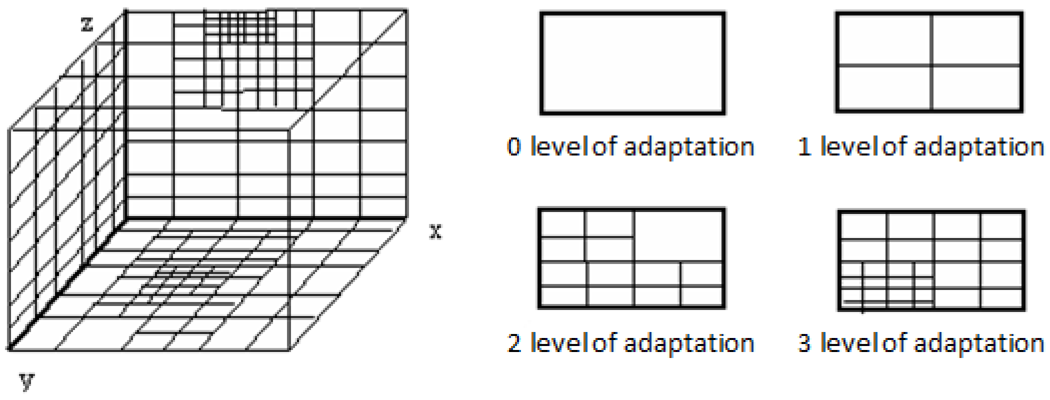


Notes:

[1] The adaptation enables resolution of small geometry details of the computational domain and high gradients of the computed values.

[2] geometry resolution allows approximation of curvilinear boundaries on a rectangular grid.

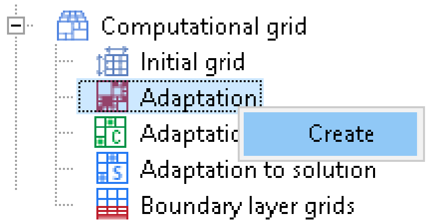
3.1.7 Adaptation of computational grid



Adaptation is splitting or merging of cells of the computational grid up to the specified level in a volume or on a surface. Splitting up n-th level includes halving the initial grid cells in each direction n times. Merging to the n-th level involves merging the cells previously split to a level m (where m>n), to the level n.

Adaptation can be specified on the surface of some **Boundary condition**,as well as on a surface or in a volume of some **Object**.

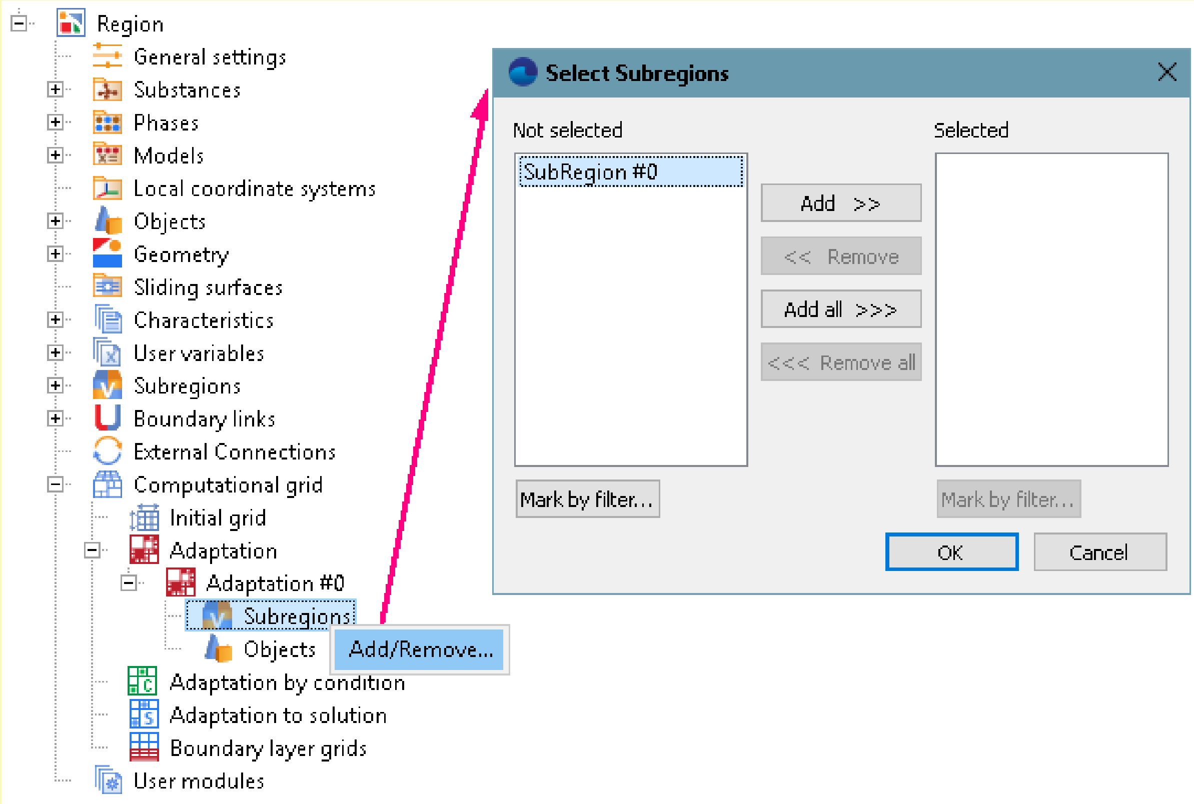
In this exercise, you must specify an adaptation on the boundary condition **Wall**. To do this, start with creation the element **Adaptation #0** in the folder **Computational grid > Adaptation** by right-clicking on the element **Computational grid > Adaptation** and selecting the **Create** command from the context menu, which opens:



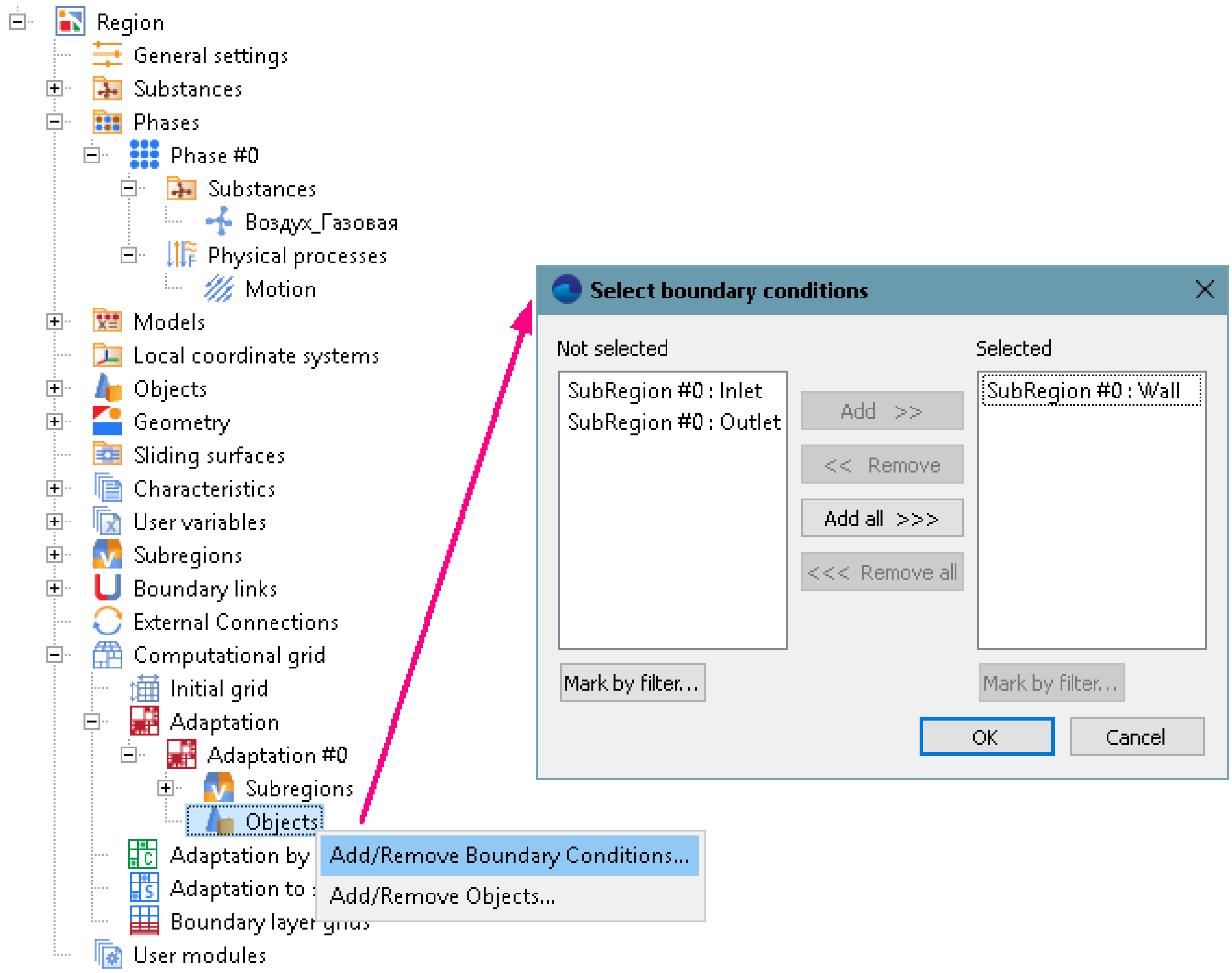
For the element **Adaptation #0** the following information is to be specified:

- a **Subregion** where the adaptation will act (in our case, the program, for your convenience, will automatically set use of the adaptation in **Subregion #0** because the project contains this **Subregion** only)
- *and* **Boundary conditions**, on which and near which the adaptation will act.

If the program does not select the desired **Subregion** automatically, where the **Adaptation** will act, select the **Subregion** manually. To do this, right-click the element **Adaptation > Adaptation #0 > Subregions** and select the **Add/Remove** command from the context menu, which opens. The **Select Subregions** dialog box will open, in which you have to place the desired **Subregion** into the pane **Selected** (using the **Add** button) and then click **OK**:



To select **Boundary conditions** you have (with right-click) open the context menu of the element **Adaptation > Adaptation #0 > Objects** and select there the **Add/Remove Boundary Conditions** command. In the **Select boundary conditions** dialog box, which opens, place the boundary condition **Subregion #0 : Wall** into the pane **Selected** and click **OK**:



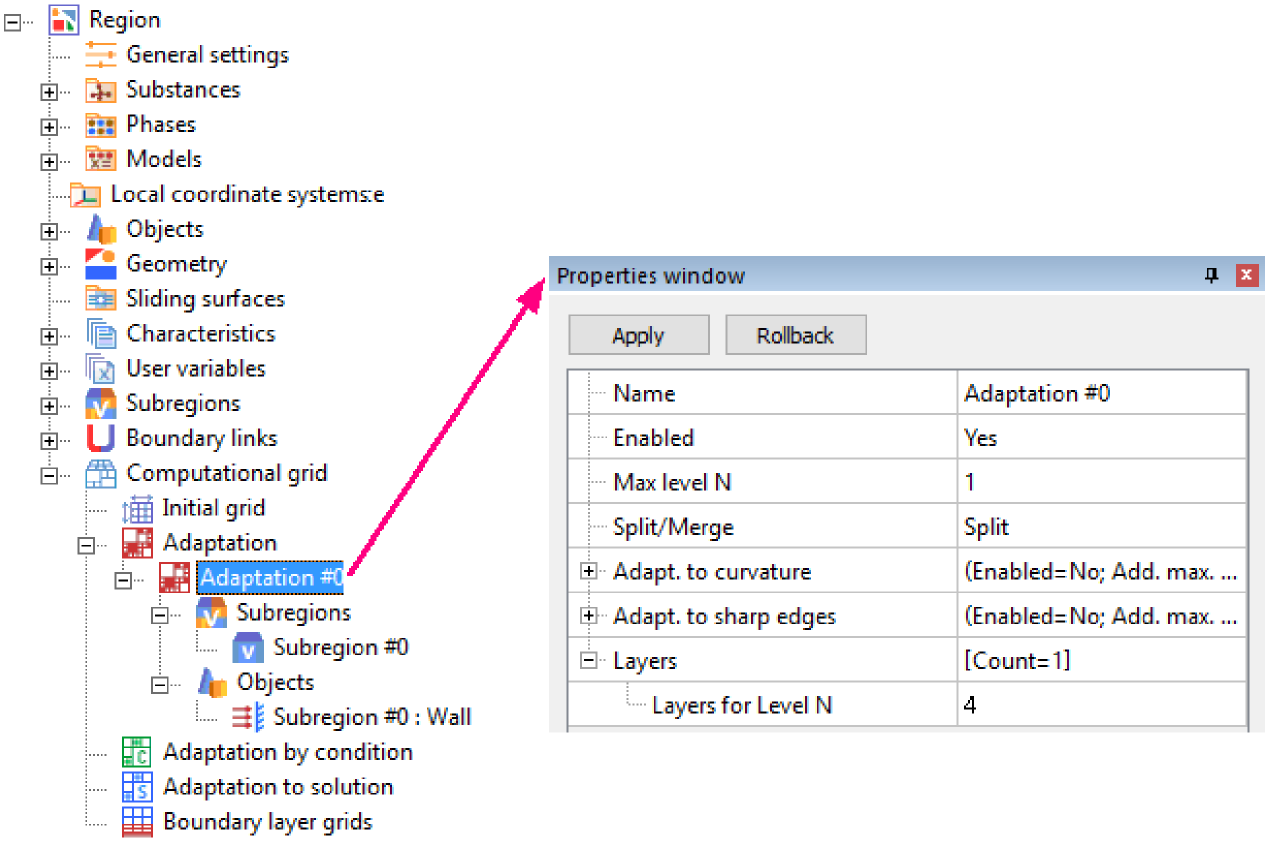
In the **Properties** window of the element **Adaptation > Adaptation #0** specify:

Enabled = **Yes**

Max level N = **1**

Layers

Layers for Level N = **4**



Note:

In order to see the computational grid with adaptation, create a layer **Computational grid** on the **Computational space**. The layer will be displayed after the calculation is done.

3.1.8 Defining control parameters of computation

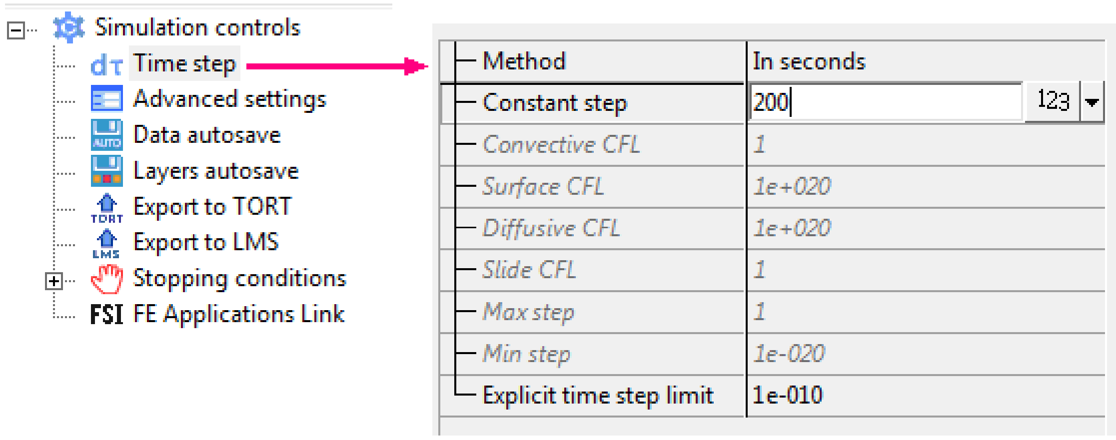
The final step in the preparation of the project is definition of **Simulation controls** (parameters that control the computation).

These settings include: time step, the selection of the scheme for approximation of the equations in space and in time, the frequency of autosave, etc.

These parameters are specified in the **Solver** tab of the project tree.

It is assumed that for most problems (except of specially specified) the user does not need to change the standard **Advanced settings**. The only thing that you need to specify is the **Time step**. In *FlowVision* There are two ways of specifying the time step: **In seconds** and **Via CFL number** (CFL number is the Courant-Friedrichs-Levy number). In our exercise you can specify the time step in seconds as the constant time step chosen by the user. In this exercise the characteristic dimension is length of the tube L. The transit time is the time required for a hypothetical particle moving with the mean flow velocity V, to pass the characteristic dimension :

$$\tau_{own} = 0.1 * \frac{L}{V} = 0.1 * \frac{2}{0.001} = 200c$$



3.1.9 Stopping conditions

Stop the computation can be done in two ways:

1. Manually using the (**Stop computation**) button
2. Specifying some **Stopping conditions**

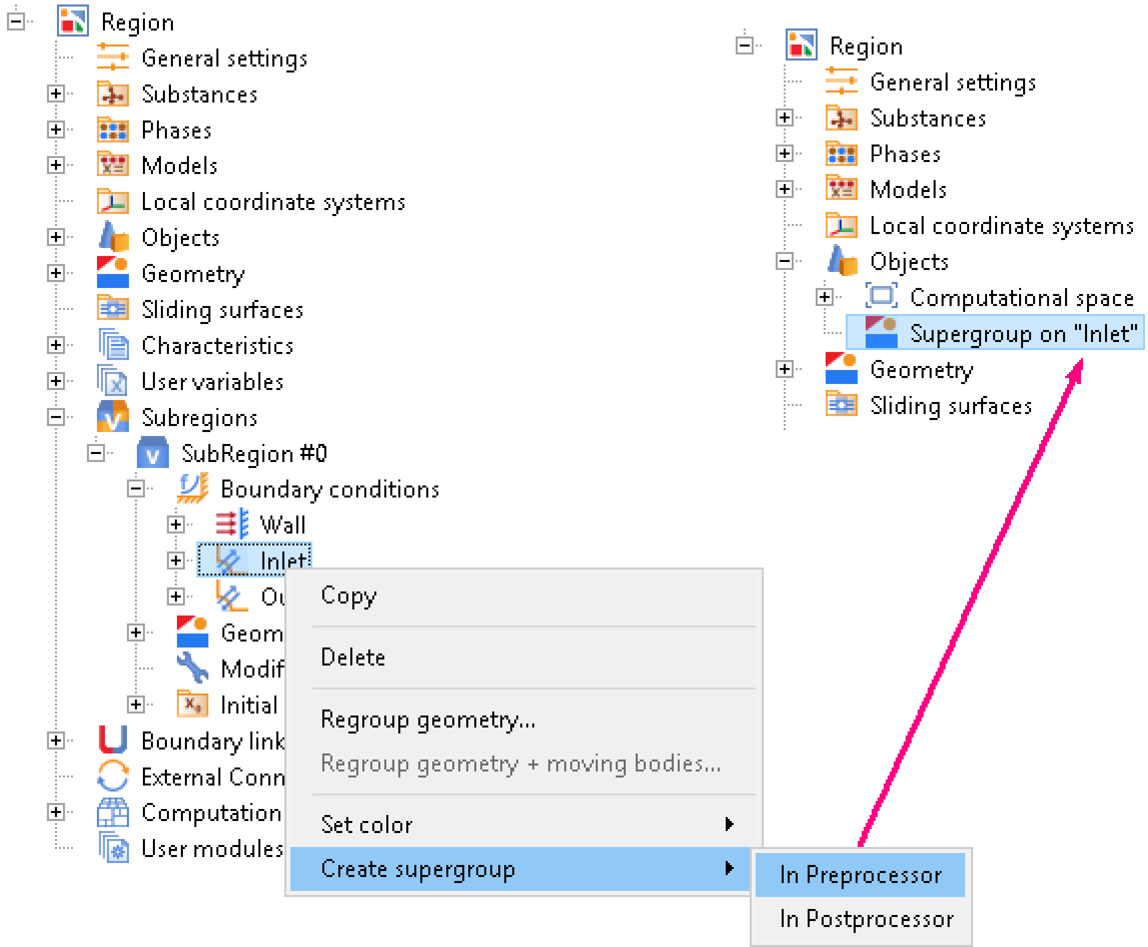
Stopping conditions are specified prior to the computation in the **Solver** tab. You can define stop by:

1. At the specified moment of time.
2. After the specified number of iterations.
3. By values of residuals of calculated variables.
4. By values of residuals of user variables (global user values or values from **Characteristics**).

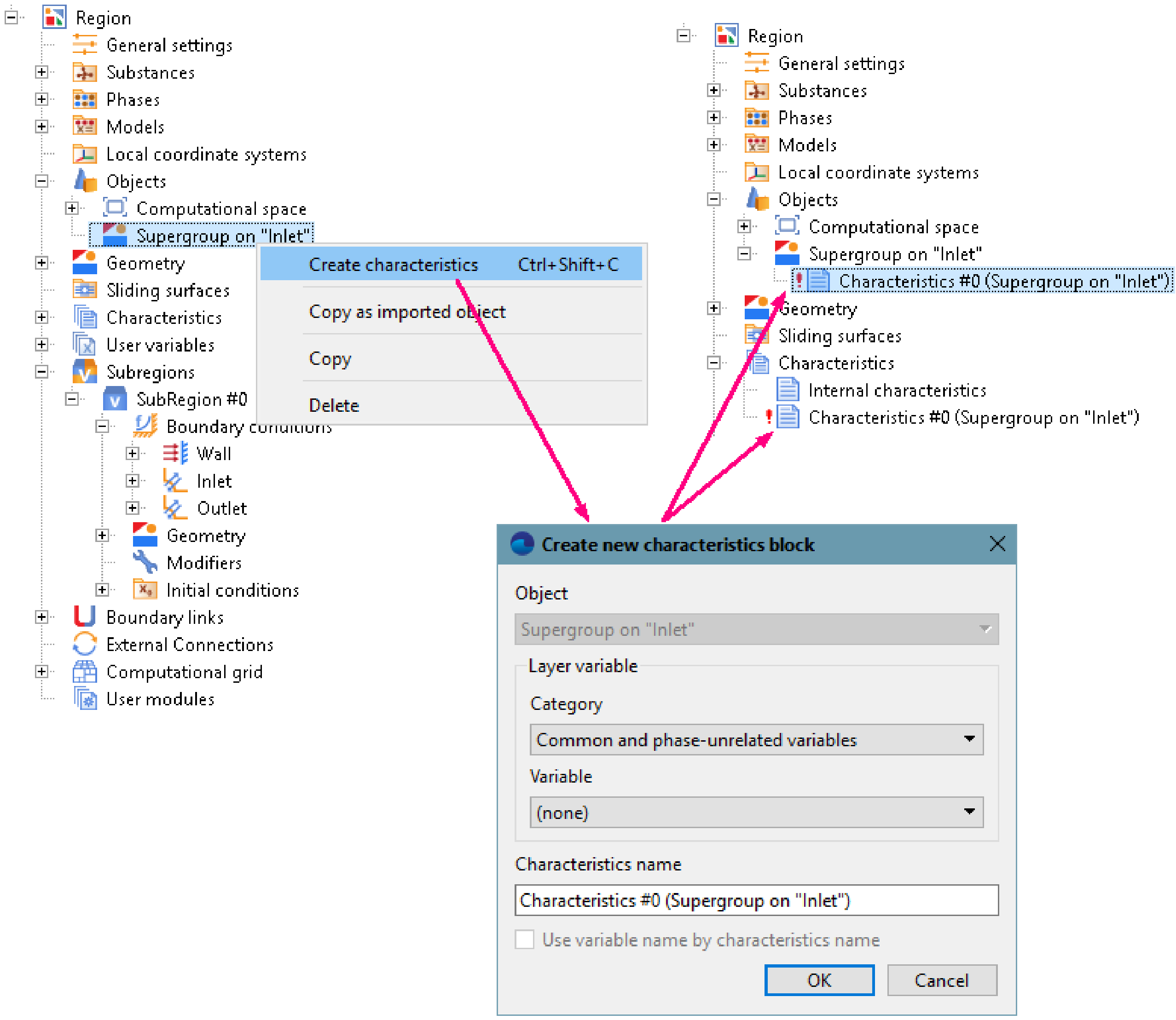
Residuals characterize the rate of changing the values over time. The smaller are the residuals, the less is the change of the evaluated values.

The **Characteristics** element provides access to information about integral values of variables in the volume or on the surface of an **Object**. We will use **Characteristics** to watch the evolution of average pressure on the inlet and *automatically stop the computation* at the moment when the average pressure stops changing.

To specify the **Characteristics**, you have firstly to create an **Object** for visualization (based on the **Inlet** boundary condition). To do this, select, in the **Preprocessor** tab, in the context menu of the boundary conditions **Inlet**, the **Create supergroup > In Preprocessor** command:

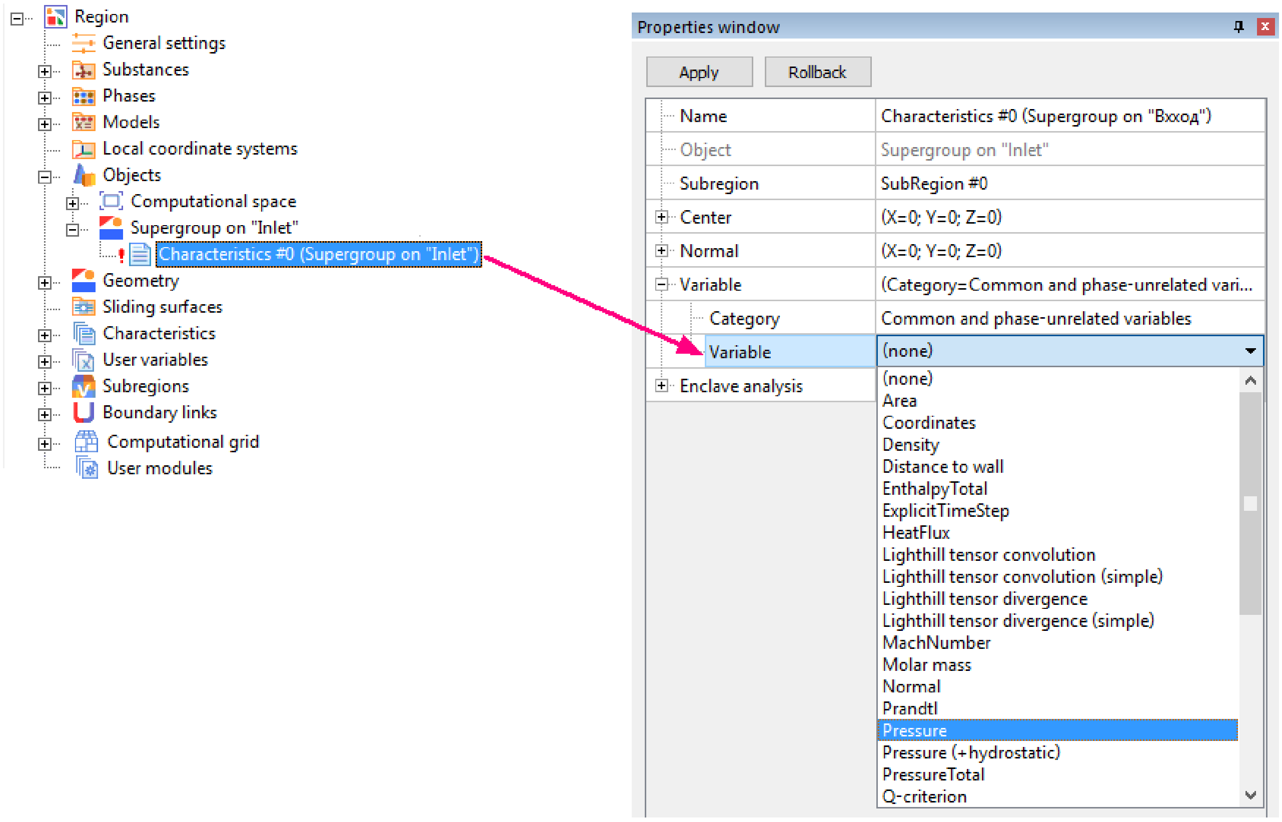


Then in the folder **Objects** in the context menu of the created **Supergroup** select the **Create characteristics** command, and then, in the **Create new characteristics block** dialog box, which opens, click **OK**:



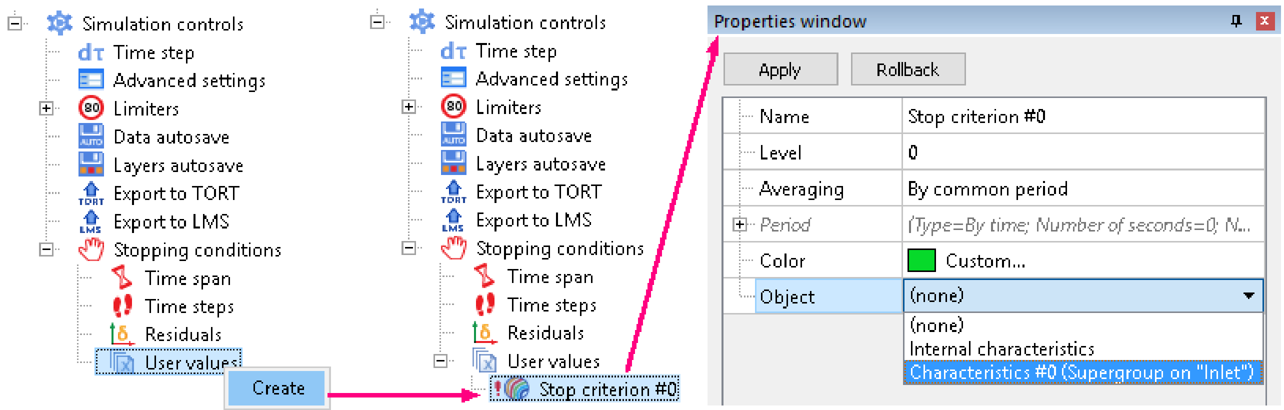
By default, no variable is specified on the **Characteristics** and so the **Characteristics** do not contain the main body of data. In order to fill the **Characteristics** with information, specify a variable in properties of **Characteristics #0**:

Variable > Variable = Pressure



In order to specify the stopping conditions:

- in the **Solver** tab, from the context menu of the folder **Stopping conditions > User values**, select **Create**:



- in the **Properties** window of the new just created **Stop criterion #0** specify:

Level	**= 0.00001**
Object	**= Characteristics #0 (Supergroup on "Inlet")**
Variable*)	**= <f surf.>**

*) This field will appear after you specify **Object = Characteristics #0 (Supergroup on "Inlet")**.

The **<f surf.>** value equals to the average value of the **Variable** selected in the **Characteristics** at the surface or in the volume of the **Object**, on which the **Characteristics** was created. In this exercise **<f surf.>** is the average **Pressure** on the surface of the **Supergroup**, which was created in the boundary condition **Inlet**.

The **Level** value is the threshold value of the residual for the selected user value (**<f surf.>**). When the residual of the user value becomes less then the specified **Level**, the computation stops.

Note:

In many further exercises stopping conditions are not set. In such cases the computation is stopped manually at any desired time by the user's decision.

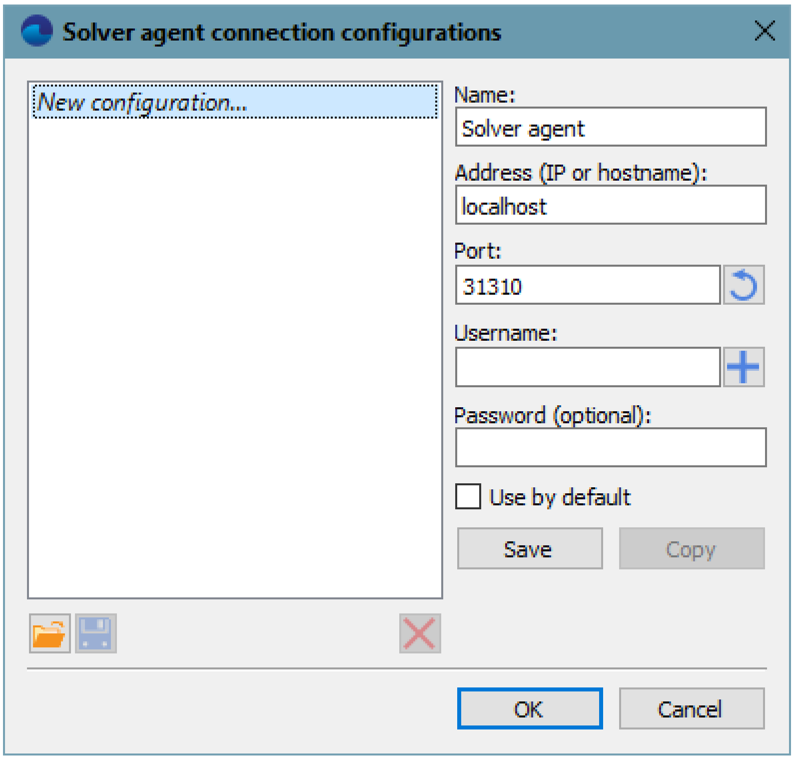
3.1.10 Starting the computation

You can start the project's computation from **Pre-Postprocessor** or from**Terminal**.

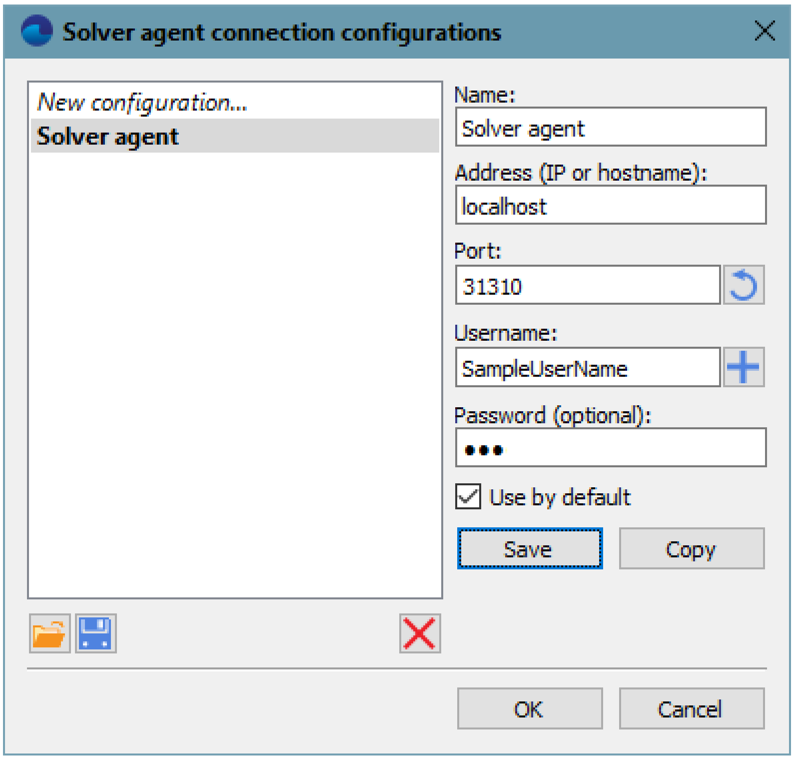
In this exercise, start from **Pre-Postprocessor** is described. **Solver-Agent** and license server are to be running.

To start the computation, you have to authorize on **Solver-Agent** (see details in the user documentation of *FlowVision* in the section *"Connection to Solver-Agent and user authentication on Solver-Agent"*). This authentication can be done automatically by the program at start of **Pre-Postprocessor**; in the case if the authentication was not done automatically, follow these steps:

- Click the (**Solver agent log in**) screen button in the **Network** toolbar.
- If the required **Configuration** for connection to **Solver-Agent** has not been set yet, set it as described in the user documentation of *FlowVision* in the section *"Connection to Solver-Agent and user authentication on Solver-Agent"*. If no **Configuration** is set, an empty **Solver agent connection configurations** dialog box will open immediately:



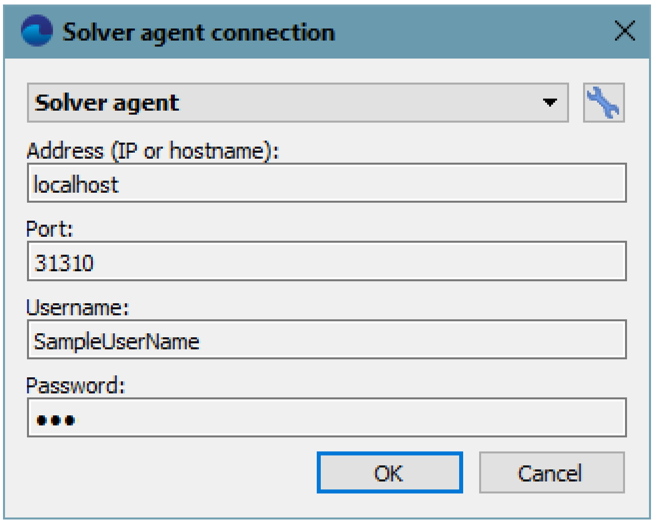
In this dialog box you have to create at least one **Configuration**. Enter data into fields (the **Password** field can be left blank), select the **Use by default** checkbox, and then click the **Save** screen button:



Then click **OK**.

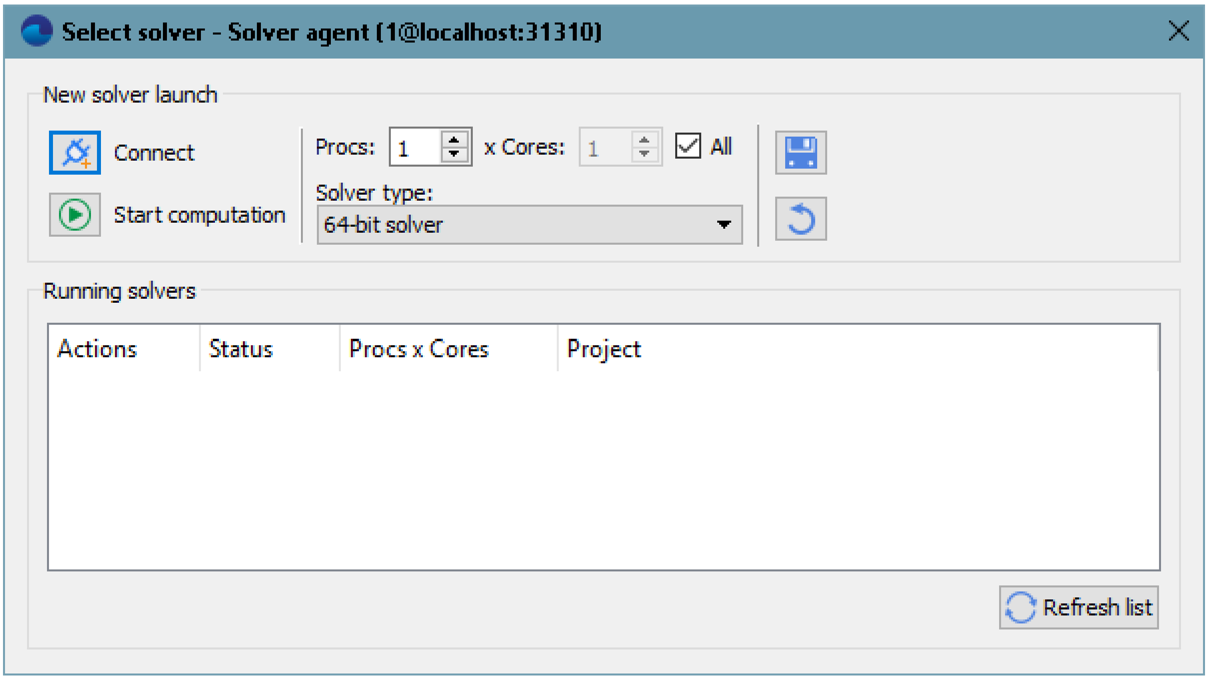
> For **Configuration**, which will be used most frequently, we recommend to select the **Use by default** checkbox to enable automatic authentication on **Solver-Agent** at each start of **Pre-Postprocessor**.
>
> If saving the password is undesired (when you need manual entering the password at each authentication on **Solver-Agent**), leave the **Password** field blank.

- The **Solver agent connection** dialog box will open where you have to select a **Configuration** for connection to **Solver-Agent**, and then click **OK**:

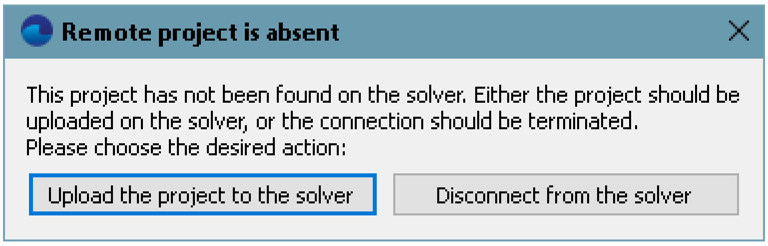


To connect to a **Solver**, you have to:

- Click the button (**Open solver selection window**) in the **Network** toolbar. The **Select solver** dialog box will open:

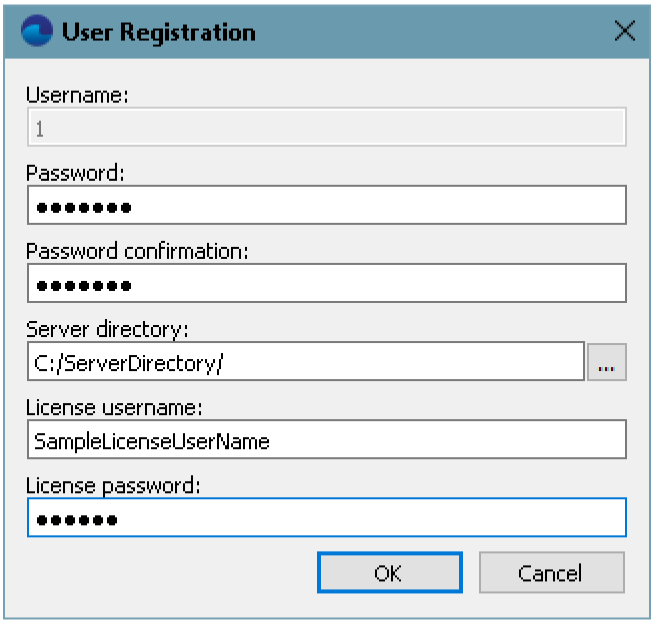


- In this dialog box click the **Start computation** screen button in the **New solver launch** group of interface elements.
- The program will prompt you to upload your project on the **Solver**:



Project files will be located in two directories: **Client** and **Server**. The **Client directory** contains the client part of the projects - the parts, which are opened in **Pre-Postprocessor**. The **Server directory** contains the server part of the project - the parts, which are loaded on the solver. The client part of the project appears when you create a project in **Pre-Postprocessor**. At the same time the **client directory** is determined. The server part of the project appears in the **Server directory** after the first upload of the project to the solver. The **server directory** is specified when a user is registered on **Solver-Agent**.

If necessary, you can open the dialog box with the **Solver-Agent**'s registration information, using the button (**Edit solver agent user information**) in the **Network** toolbar after authorization on **Solver-Agent**.



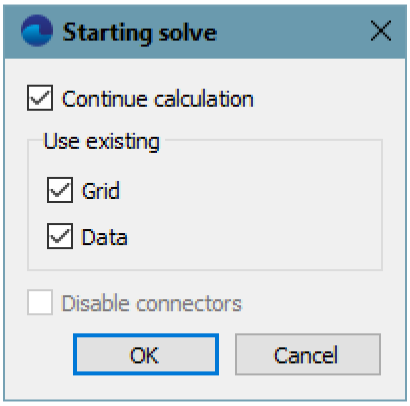
Client and **Server directories** must *not*:

- be the same
- be nested to each other
- be the same as the installation directory
- be nested to the installation directory.

The computed results are stored in server part of the project. You can see the results of the computation in **Pre-Postprocessor** only if the project is opened in **Pre-Postprocessor**, **Pre-Postprocessor** is connected to a **Solver** synchronized with the **Solver**.

If necessary, the **Server directory** can be changed. However, you should remember that after that change all the results of computations that were in the old **Server directory** will no longer be available after connection of **Pre-Postprocessor** to **Solver** unless you have manually copied them to new directory.

Then the **Starting solve** dialog box will open:



Click there **OK**, this will start the project's computation.

After this you can stop and resume the computation using buttons (**Stop computation**) and (**Start computation**) in the **Network** toolbar.

Before starting the computation we recommended that you specify **Visualization** to see the results of computation in dynamics.

3.1.11 Visualization

Visualization of the results of the computation is performed in **Postprocessor**.

*Visualization of the computation results are only available if the project is opened in **Pre-Postprocessor** and loaded on a **Solver**.*

***Pre-Postprocessor** and **Solver** must have connection between them.*

Postprocessor provides the user with the following capabilities:

1. A set of standard methods of flow visualization in the **View** window
2. Interactive tuning of visualization parameters
3. Displaying integral values of variables in the **View** window
4. Indication of the visualization method in a separate **Information** window

To display the results of computation in **Postprocessor** you have to:

- create and configure the corresponding object
- create a layer or characteristics of required type on the object
- select a variable and define the necessary settings of the layer or characteristics

Elements for visualization the results of the computation should be chosen depending on the type of data, which are to be displayed:

Data Type	Name of the element
Integral values:	
Value of a variable on a surface and in a volume	**Characteristics**
Local values:	
Local values of a scalar variable along a line	**Plot along line** **Plot along curve** **Plot along ellipse**
Local values of a scalar variable at a surface	**Color contours**
Local values of a scalar variable in a volume	**Isosurface**
Local values and directions of vector variable on a surface or in a volume	**Vectors** **Streamlines**

For long-term computations we always recommend you to visualize data during the computation, as in this case you can permanently control over the process of convergence of the solution and, if necessary, intervene in the process of computation when the solution becomes numerically unstable.

3.1.11.1 Charachteristics (pressure variation)

To display information about the value of the pressure on inlet, you can use the element **Characteristics**, created earlier to define a **Stop criterion**. Information from **Characteristics** is displayed in the **Info** window and recorded into a text file. Recording the information into a text file allows you to monitor changing the variable over time.

In order to record the contents of the **Characteristics** into the file, open the tab **Postprocessor** and specify in the **Properties** window of **Characteristics #0**:

Save to file

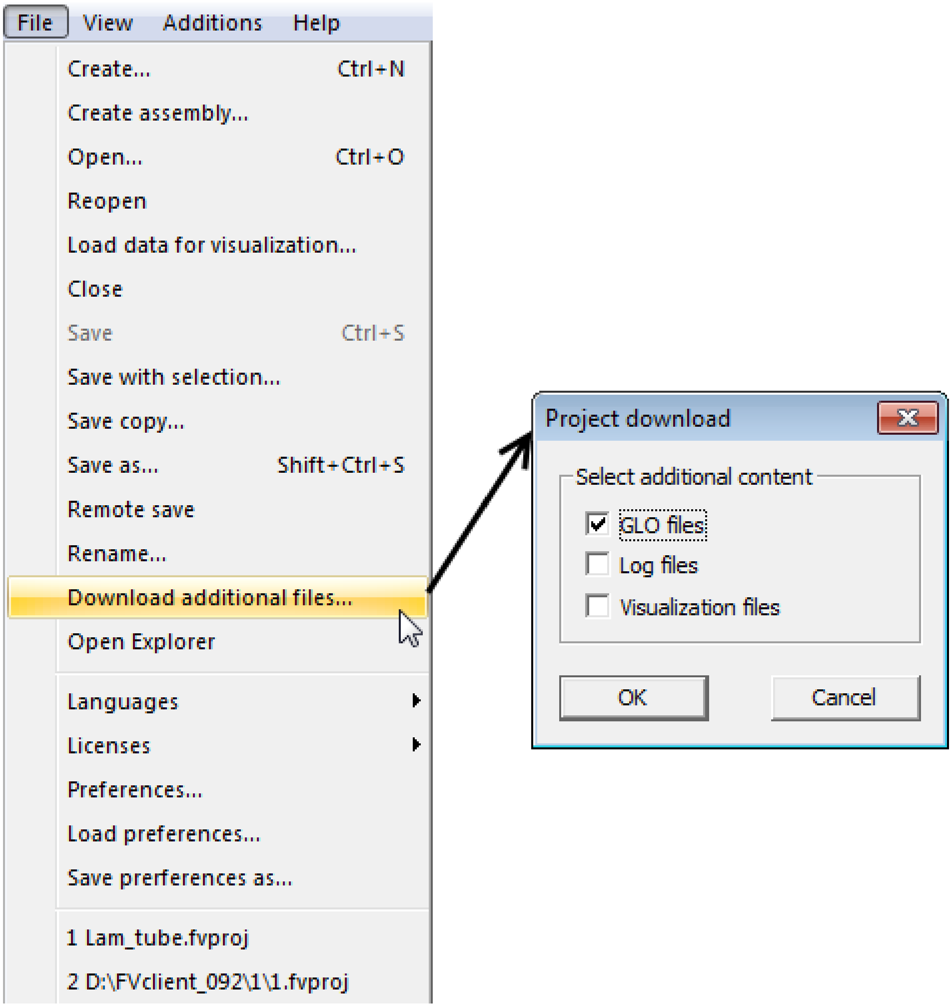
  Type = Automatic

After the computation is done, the **Characteristics** become informative.

To open the **Info** window, select **Characteristics #0** in the project tree in **Pre-Postprocessor** and click on the button  (**Show info window for selected object**) in the **Work modes** toolbar*).

All data from the **Info** window is stored in a text `glo`-file, which can be exported into *Excel* to plot the dynamics of values variation. By default the `glo`-file is saved into project directory in the server directory of the user. In order to download the `glo`-file from the server directory to the client directory, do the following:

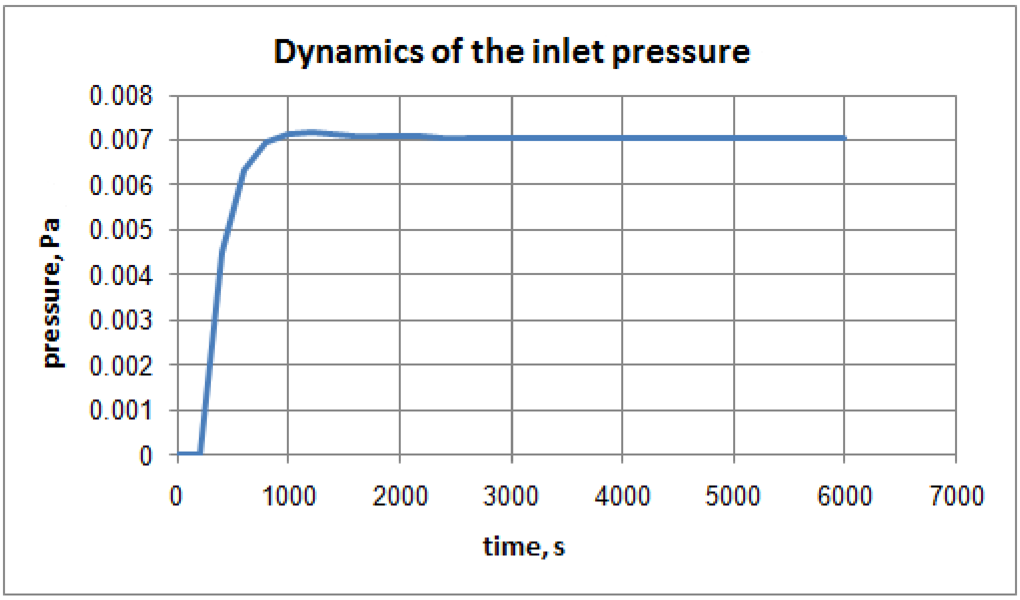
- in the **File** menu, select **Download additional files**
- in the **Project download** window, which opens, select **GLO files**



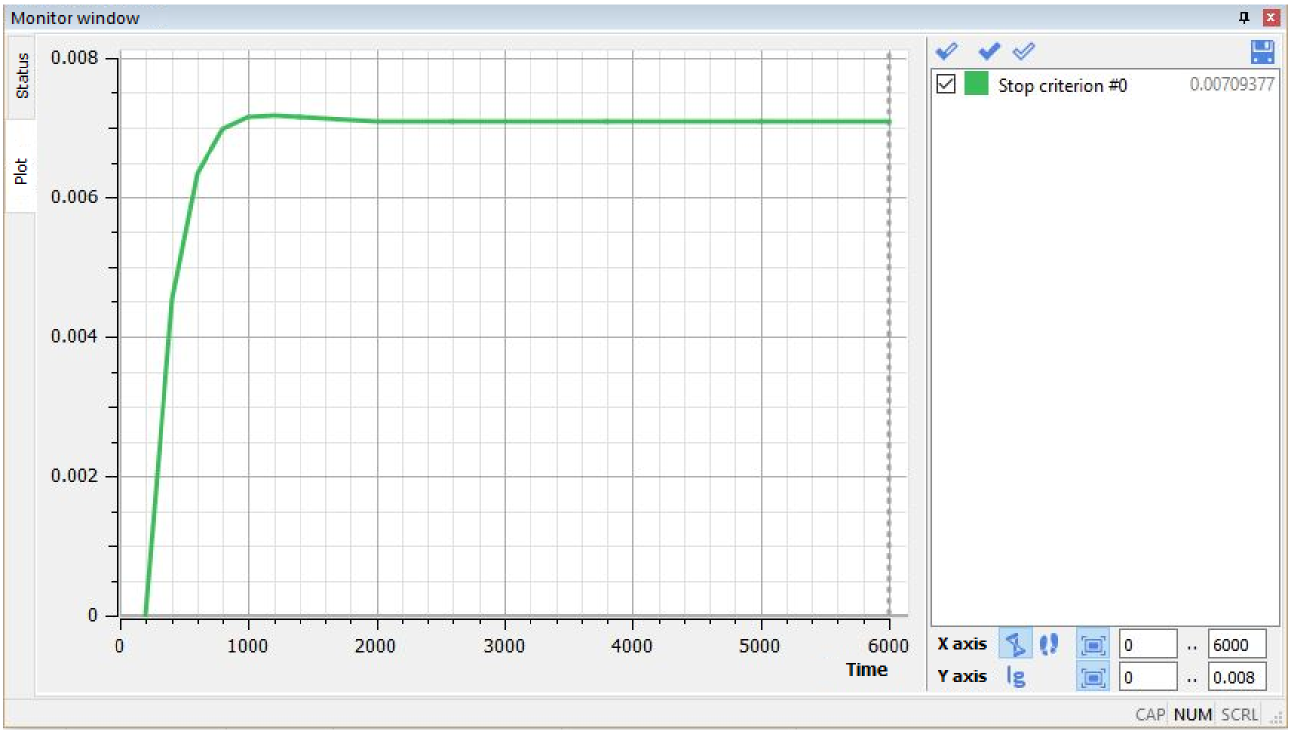
To start *Windows Explorer*, which opens the client directory of the current project, use the menu command **File > Open Explorer**.

In order to plot the variation of the pressure on inlet over time in any third-party graphics editor (*Microsoft Excel*, *Grapher*, etc.), do the following steps:

- open or import the recorded `glo`-file in the appropriate graphical editor
- plot variation of **Avg**$^{*)}$ over **Time**



Since a **Stop criterion** was created based on the **<f surf.>** *) variable, its variation can be displayed directly in **Plot** tab of the **Monitor** window of **Pre-Postprocessor**:



Note:

*) Variable **<f surf.>** in the **Info** window and in the **Stop criterion** and variable **Avg** in the `glo`-file correspond to the average value of **Pressure** on **Inlet**.

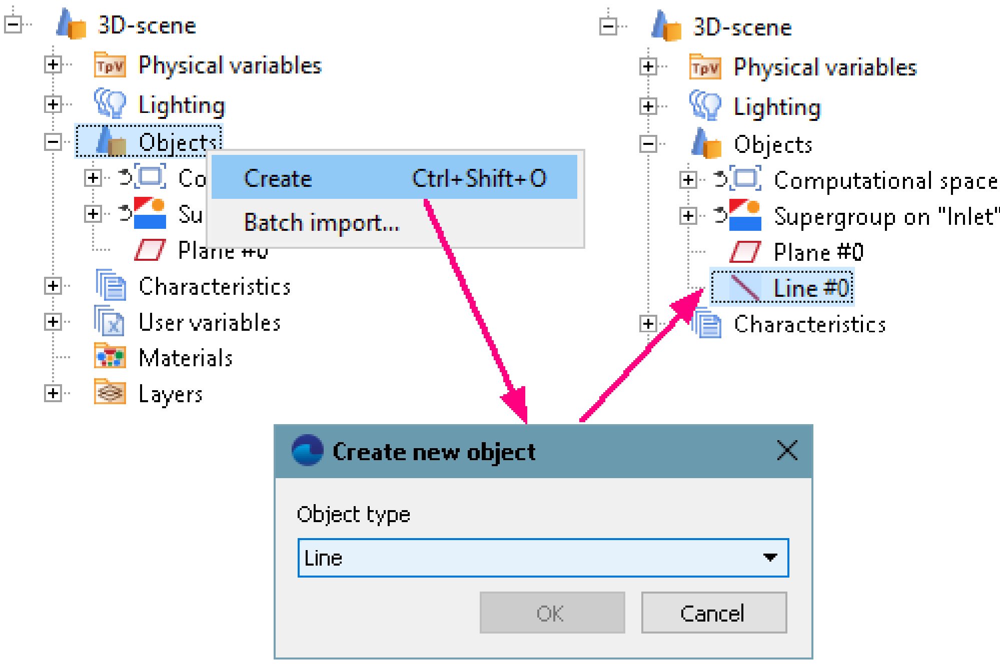
3.1.11.2 Plot along line (pressure distribution)

The layer **Plot along line** allows you to display the distribution of the scalar variable along the selected line as a plot.

In this exercise, we will use the layer **Plot along line** on a **Line** object to visualize the distribution of pressures along the tube axis.

Create in **Postprocessor** a **Line** object for visualization:

- from the context menu of the folder **Objects** in **Postprocessor** select **Create**
- the **Create new object** dialog box will open, select there **Object type** = **Line**



In the **Properties** window of the created object **Line #0** specify:

Object

Reference point

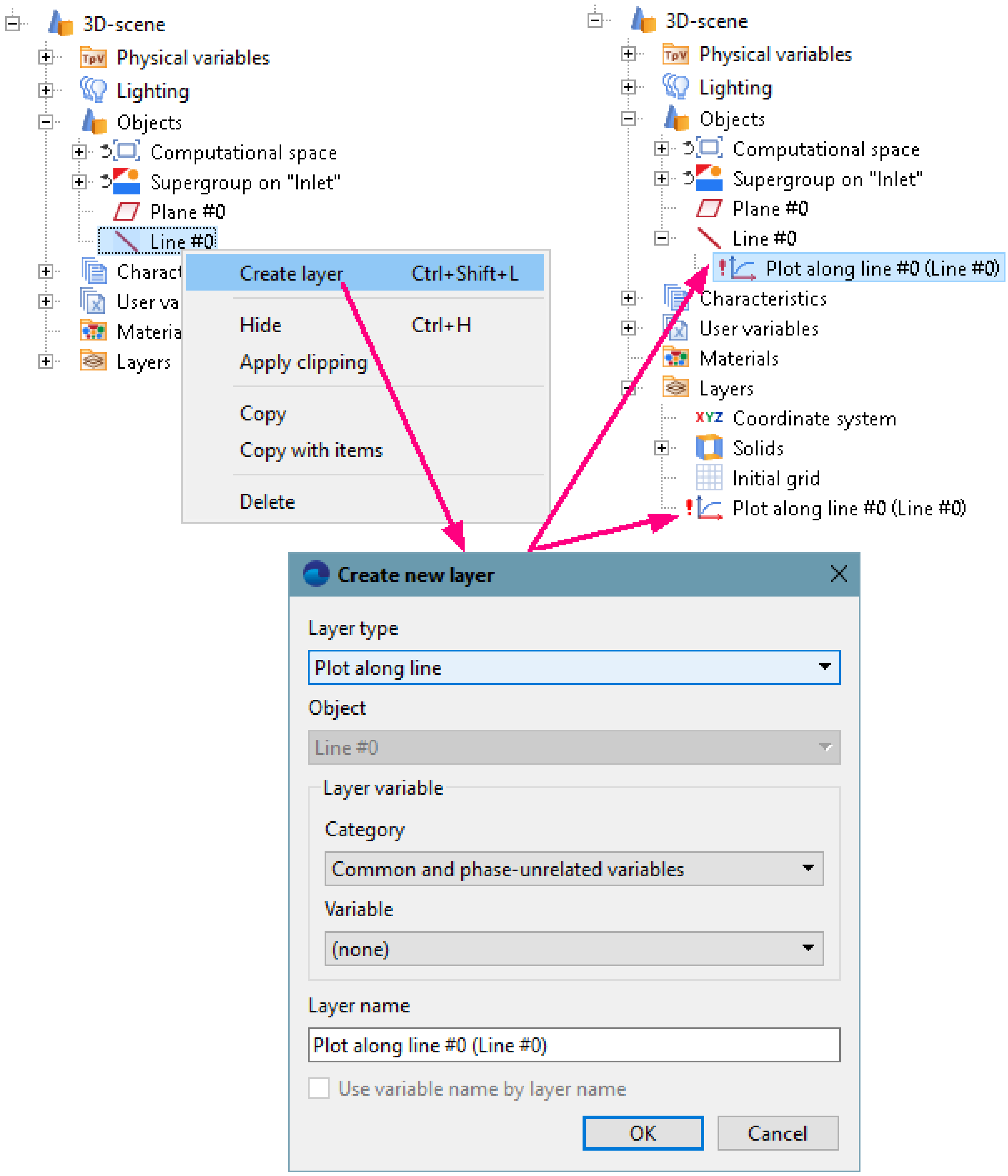
X	**0**
Y	**0**
Z	**0.001**

Direction

X	**0**
Y	**0**
Z	**1**

Create in **Postprocessor** a layer **Plot along line** on **Line #0**:

- in the context menu of **Line #0** select **Create layer**
- specify **Layer type** = **Plot along line**



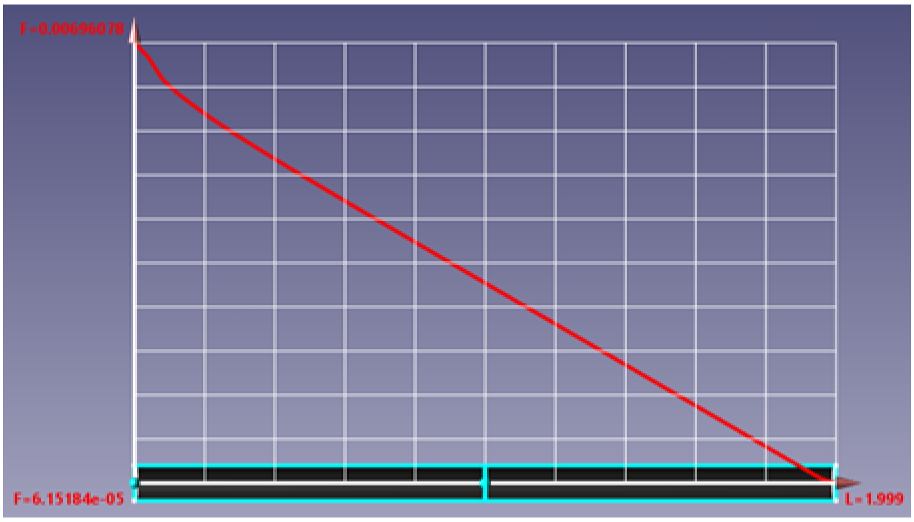
In the **Properties** window of the layer **Plot along line**, specify:

Variable

Variable	**Pressure**

In order to increase the number of points on which the plot is built, specify:

Number of points = 100



3.1.11.3 Vectors (velocity distribution)

The layer **Vectors** visualize a vector field. The direction of the vector coincides with the direction of the vector field at the starting point of the vector and the vector length is proportional to the modulus of the field at this point.

In this exercise, we will use the layer **Vectors** on a **Plane** in order to visualize the vector field of velocity in the plane of the flow.

In order to the layer do not obscure the geometry, specify:

- in properties of **Plane #0** in the **Postprocessor** specify:

  Clipping object = Yes

- in properties of the layer **Solids** specify:

  Clipped = Yes

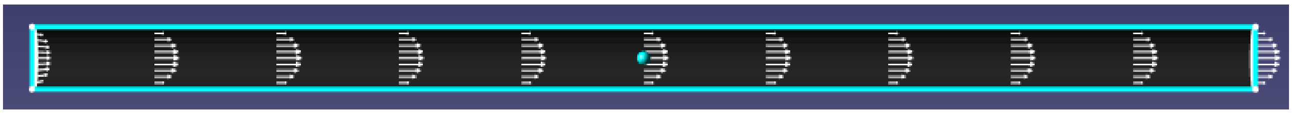
Create a layer **Vectors** in the **Plane #0**:

- in the context menu of the **Plane #0** select **Create layer**
- at creating the **Layer**, specify **Layer type = Vectors**

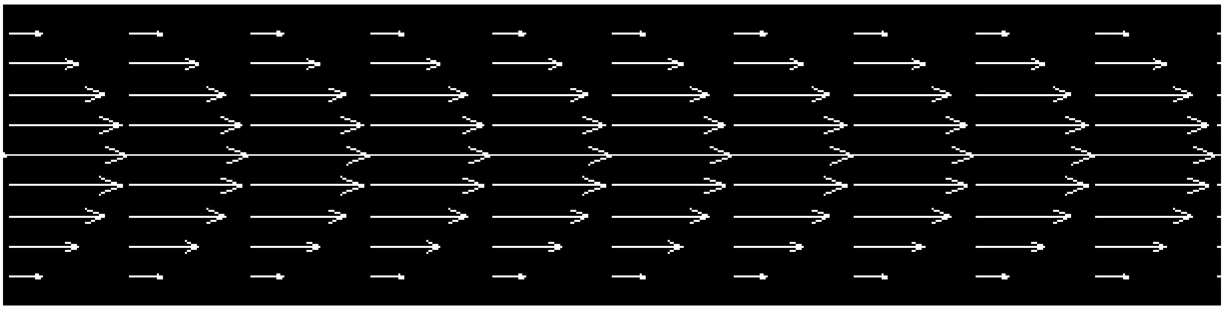
In the**Properties** window the program will automatically specify, what a variable will be used to build the **Layer**:

Variable > Variable = Velocity

In the **View** window a visualization of the velocity distribution in the plane of the flow will appear:



To increase number of vectors in the direction along the tube, specify in properties of the layer: **Grid > Size 2 = 50**.

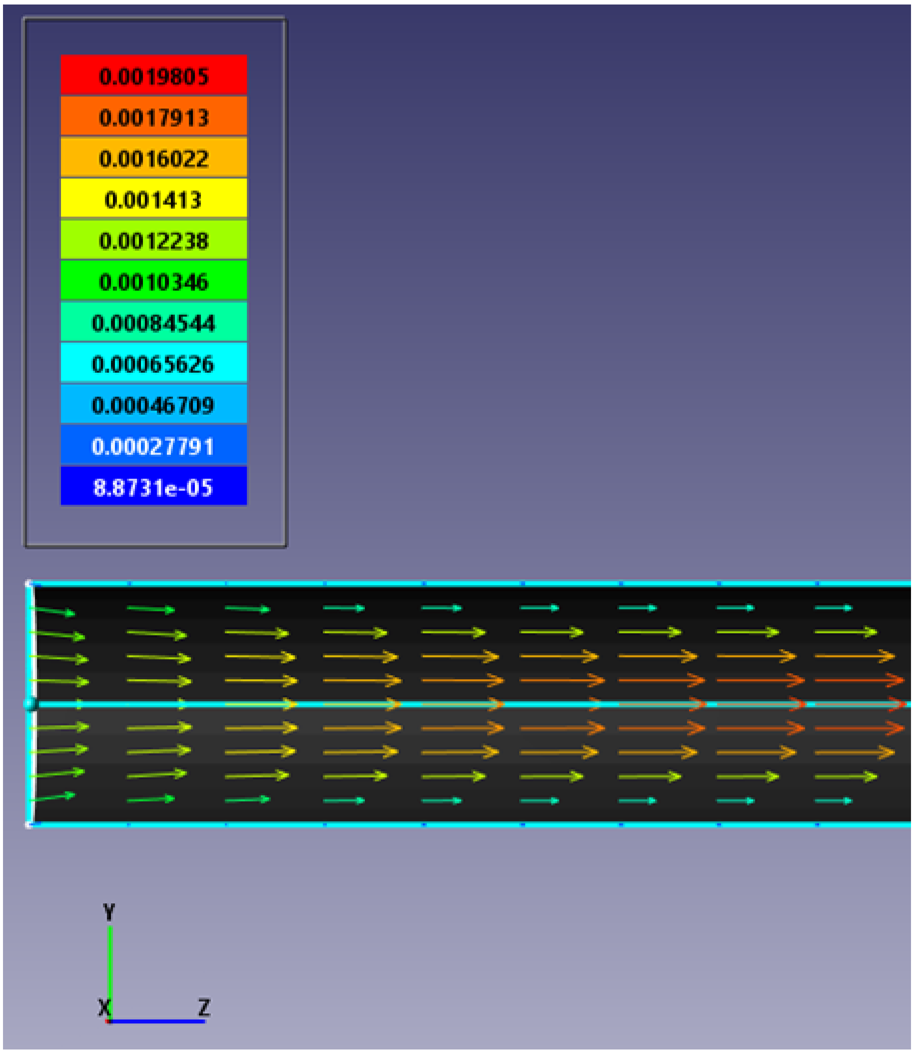


(by default, in properties of the layer the following parameters were set: **Grid > Size 1 = 11** and **Grid > Size 2 = 11**)

To paint the vectors with their absolute values and display the legend to this palette, specify in properties of the layer:

Coloring > Variable > Variable = Velocity

Coloring > Palette > Appearance > Enabled = Yes
Coloring > Palette > Appearance > Style = Style 1



3.1.11.4 Color contours (distribution of velocity's modulus)

The **Color contours** layer visualizes the distribution of a scalar variable using color transitions.

This example illustrates use of a layer **Color contours** for visualization of the value of velocity in the plane of the circular cross section of the tube.

In **Postprocessor** create a **Plane** for visualization:

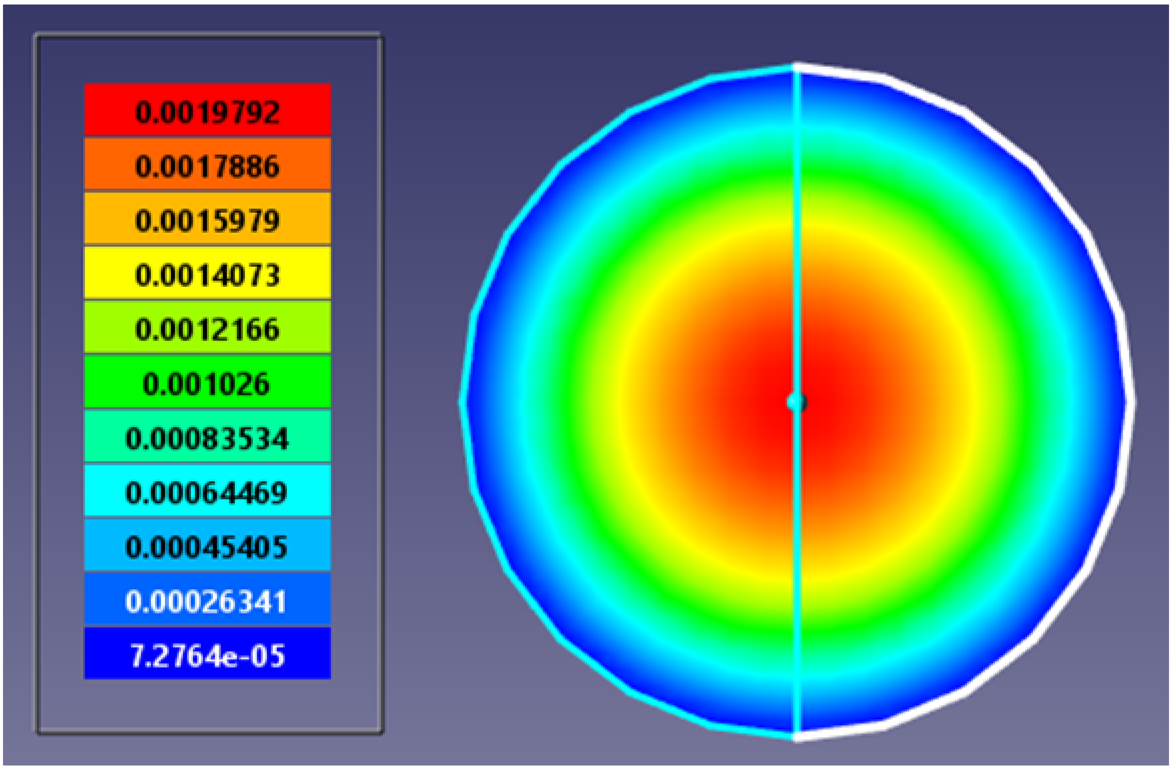
- from the context menu of the folder **Objects** in **Postprocessor** select **Create**
- in the **Create new object** dialog box select **Type** = **Plane**

In the **Properties** window of the new just created **Plane #1**, click on the button to direct the plane's normal along axis Z.

Create a layer **Color contours** on the **Plane #1**:

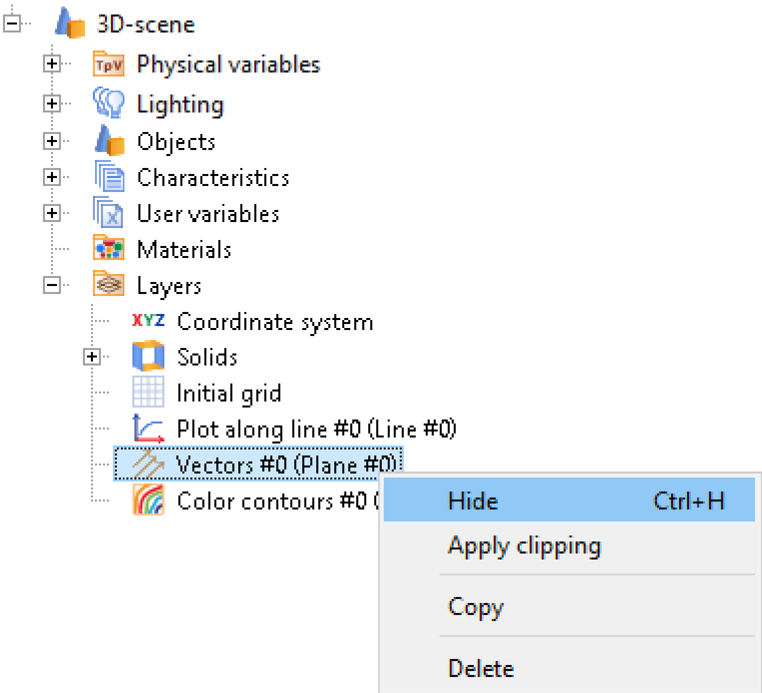
- in the context menu of the **Plane** select **Create layer**
- specify **Layer type = Color contours**

In properties of the new **Color contours** layer specify:
Variable > Variable = Velocity
Palette > Appearance > Enabled = Yes
Palette > Appearance > Style = Style 1



Note:

If you wish to hide some layers or objects displayed in the **View** window, use the **Hide** command from their context menus:



4 Physical processes

The examples of this chapter demonstrate how to model:

- Motion of fluid
- Heat transfer
- Turbulence
- Mass transfer
- Free surface
- Dispersed media
- Radiation
- Electrodynamics

4.1 Motion of fluid

In order to simulate the laminar flow of the liquid, it is necessary:

- In properties of the **Substance** specify **Aggregative state** = **Liquid**.
- In properties of the **Substance** you have to specify values of the **Molar mass**, **Density**, **Specific heat capacity** and, if necessary, the **Viscosity**.
- Enable computations of equations of **Motion**.
- Specify the appropriate initial and boundary conditions for the velocity and pressure.

In order to simulate the laminar flow of gas at Mach number less than 0.3, it is necessary:

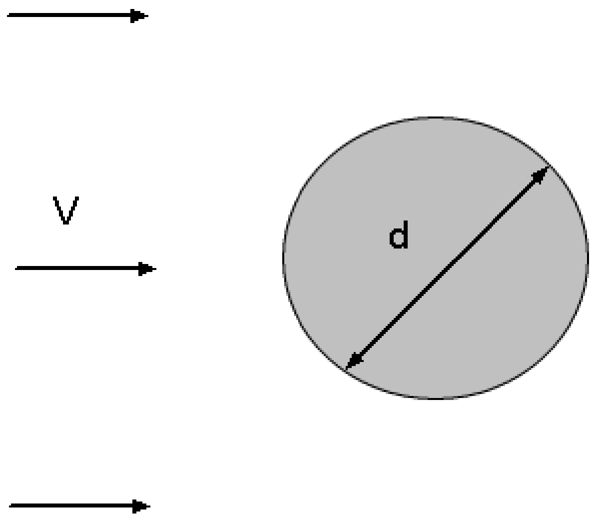
- In properties of the **Substance** specify **Aggregative state** = **Gas**.
- In properties of the **Substance** you have to specify values of the **Molar mass**, **Density**, **Specific heat capacity** and, if necessary, the **Viscosity**.
- Enable computations of equations of **Motion**.
- Specify the appropriate initial and boundary conditions for the velocity and pressure.

In order to simulate the laminar flow of gas at Mach number greater than 0.3, it is necessary:

- In properties of the **Substance** specify **Aggregative state** = **Gas**.
- In properties of the **Substance** you have to specify the computation of the **Density** by the ideal gas law, the **Molar mass** and **Specific heat capacity**, and, if necessary, the values of **Viscosity** and **Thermal conductivity**.
- Enable calculation equations of **Motion** and **Heat transfer**.
- Specify the appropriate initial and boundary conditions for the velocity and pressure. In the simulation of flow around bluff bodies it is desirable to specify some initial conditions around them corresponding to the parameters of flow deceleration.
- When simulating movement at Mach numbers greater than 1 it is recommended to specify constraints for the computation.

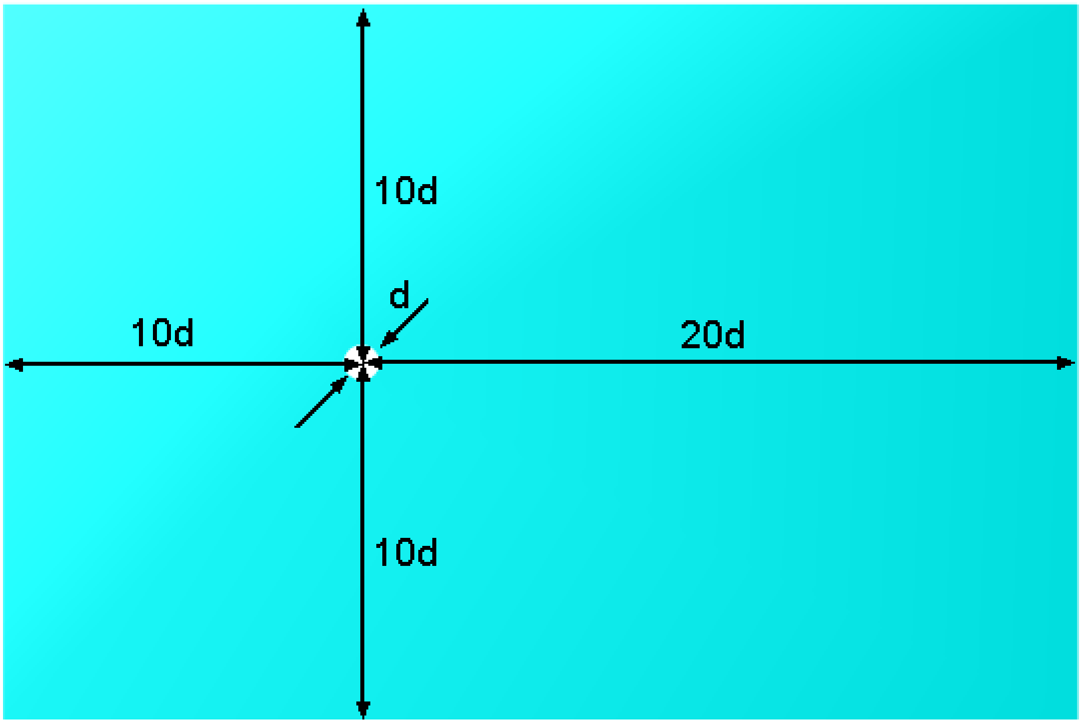
4.1.1 Laminar flow around circular cylinder

In this exercise we consider an external flow around a cylinder by a laminar flow. As we are interested in parameters of the flow in the cross section only, let us assume that length of the cylinder is infinite. Using this approach, it is possible to simulate the problem in a 2D setting.

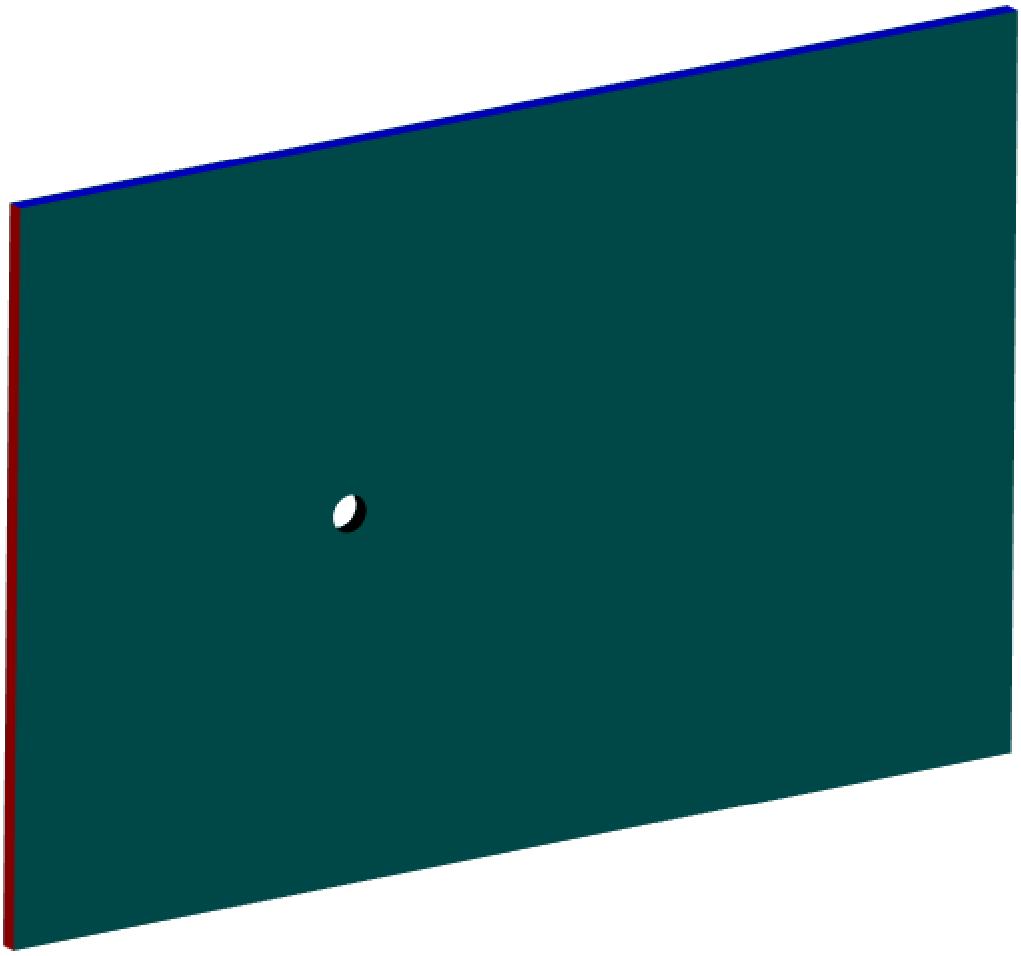


Dimensions:			
Cylinder diameter	d	= 0.02	[m]
Flow parameters:			
Velocity	V	= 0.008	$[m \cdot s^{-1}]$
Substance properties:			
Density	ρ	= 1.25	$[kg\ m^{-3}]$
Viscosity	μ	$= 2\ 10^{-5}$	$[kg\ m^{-1} \cdot s^{-1}]$
Reynolds number:	$\mathrm{Re} = \frac{Vd\rho}{\mu} = 10$		
Geometry:	**`Cylinder_lam.wrl`**		
Project:	**`Cylinder_lam`**		

4.1.1.1 Computational domain



The geometry model for simulating external flow is a limited spatial area around a material body. Sizes of the area are specified depending on specifics of the problem. For example, when external subsonic (the Mach number is below 1, M<1) flow around a material body is simulated, it is recommended to place outer boundaries of the computational domain at a distance at least 10 times more then specific body size so that flow disturbances, which appear near the body, do not reach limits of the computational domain.



Computation in *FlowVision* is always carried out in three-dimensional geometry space. So you can specify some small thickness of the geometry model to simulate the problem in 2D setting. To do so when you prepare the geometry in a CAD system, you have to:

1. create a two-dimensional sketch of the geometry of the computational domain in the plane XY
2. specify a constant thickness of the geometry in the direction Z (apply extending along this direction).

The geometry model of the computational domain, which is fully prepared to the further work, is stored in the file `Cylinder_lam.wrl`.

4.1.1.2 Physical model

Specifying the physical model (**Substances**, **Phases**, and **Models**) is done in the **Preprocessor** tab. Follow these steps there:

In the folder **Substances:**

- Create **Substance #0**
- Specify the following properties of the **Substance #0**:

Aggregative state **= Gas** *)

Molar mass

Value	= 0.0289	[kg $mole^{-1}$]

Density

Value	**= 1.25**	[kg m^{-3}]

Viscosity

Value	**= 2e-5**	[kg m^{-1} s^{-1}]

Specific heat

Value	**= 1009**	[J kg^{-1} K^{-1}]

In the folder **Phases:**

- Create a continuous **Phase #0**.
- In **Phase #0** add **Substance #0** into the folder **Substances**
- Specify in properties of the folder **Phase #0 > Physical processes**:

Motion	**= Navier-Stokes model**

In the folder **Models**:

- Create **Model #0**
- Add **Phase #0** into subfolder **Model #0 > Phases**
- Specify in the folder **Init. data > Init. data #0**:

Velocity (Phase #0)

Value > X	**= 0.008**	[m s^{-1}]

Note:

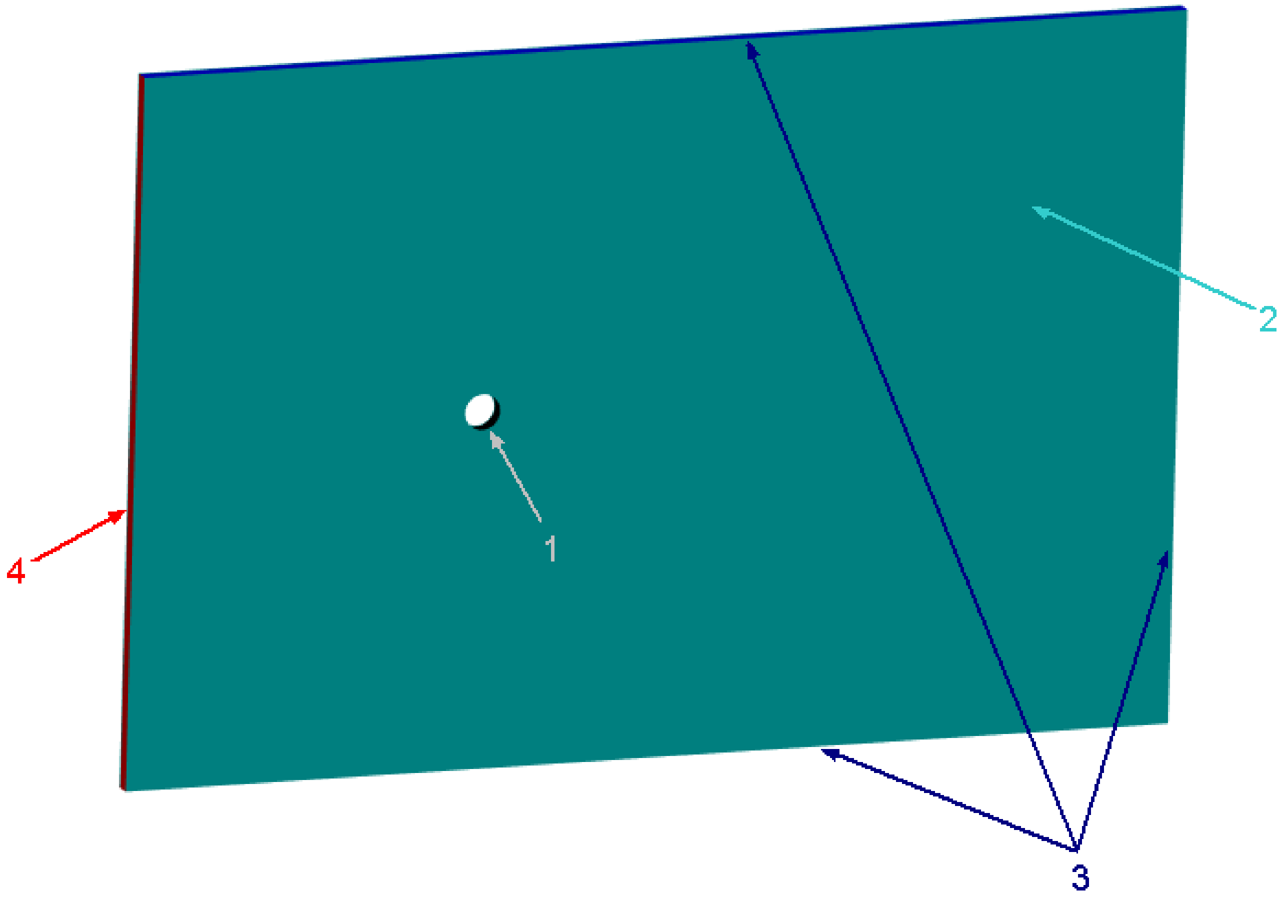
*) A gas moving with speed V < 0.17 M (which has the Mach number less then 0.17), behaves as an incompressible liquid with constant density, so the simulation of its motion can be defined with either **Aggregative state = Gas** or **Aggregative state = Liquid**.

4.1.1.3 Boundary conditions

In the **Properties** window of the **SubRegion #0**, specify:

Model = Model #0

Specify the following boundary conditions:



Boundary 1

Name **= Wall**

Type **= Wall**

Variables

Velocity (Phase #0) **= No slip**

Boundary 2

Name **= Symmetry**

Type **= Symmetry**

Variables

Velocity (Phase #0) **= Slip**

Boundary 3

Name **= Outlet**

Type **= Free Outlet**

Variables

Velocity (Phase #0) **= Pressure**

Value **= 0** [Pa]

Boundary 4

Name **= Inlet**

Type **= Inlet/Outlet**

Variables

Velocity (Phase #0) **= Normal mass velocity**

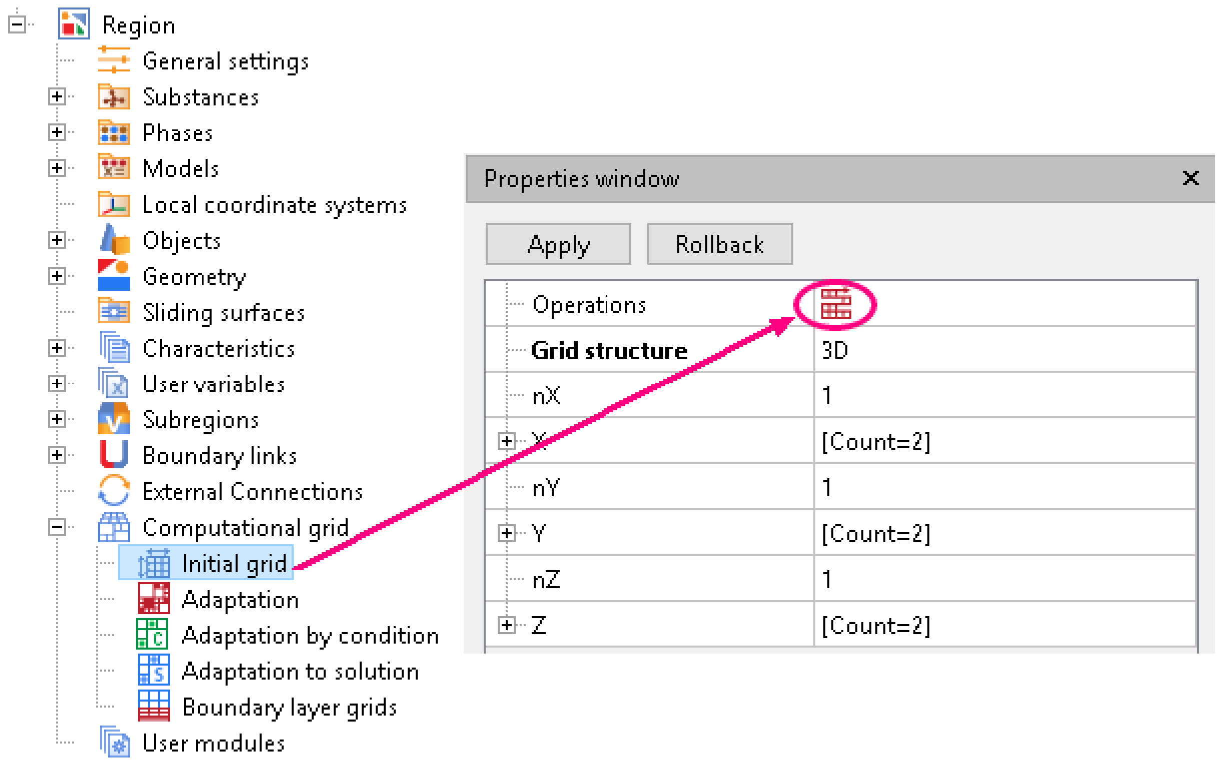
Specify the numerical value of the **Mass velocity**:

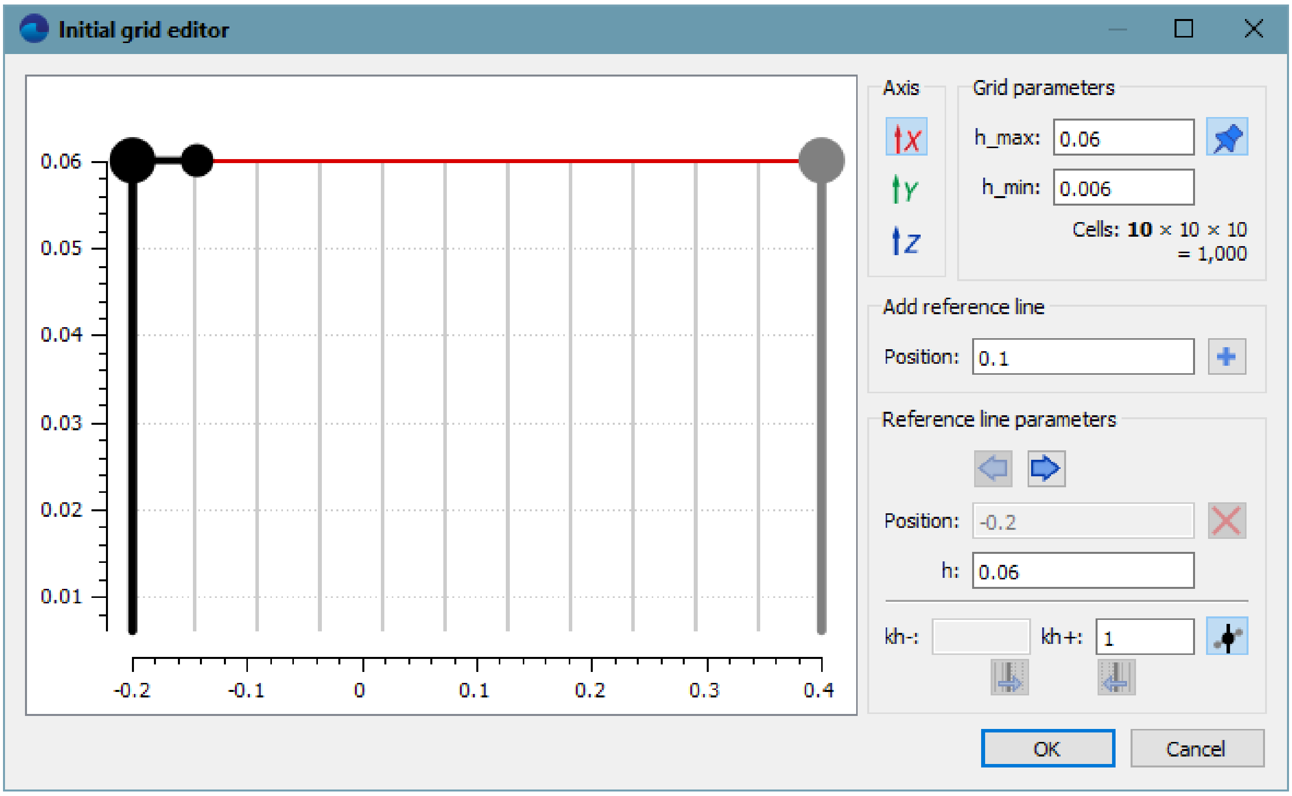
Velocity (Phase #0)

Mass velocity = **0.01** [kg m^{-2} s^{-1}]

4.1.1.4 Initial grid

In this example, in order to better resolve the flow near the cylinder, it is necessary to specify a two-dimensional non-uniform initial computational grid, condensed near surface of the cylinder. To build such grid, use the **Initial grid editor**, which is called by the button from the **Properties** window of the element **Computational grid > Initial grid** of the project tree:





As simulating of the flow will be done in a 2D setting, parameters of the flow are calculated only in two directions (X and Y), and along the third direction (Z) the computational domain will be resolved only by one cell and parameters of flow will not change along Z.

Creating a non-uniform grid with condensing

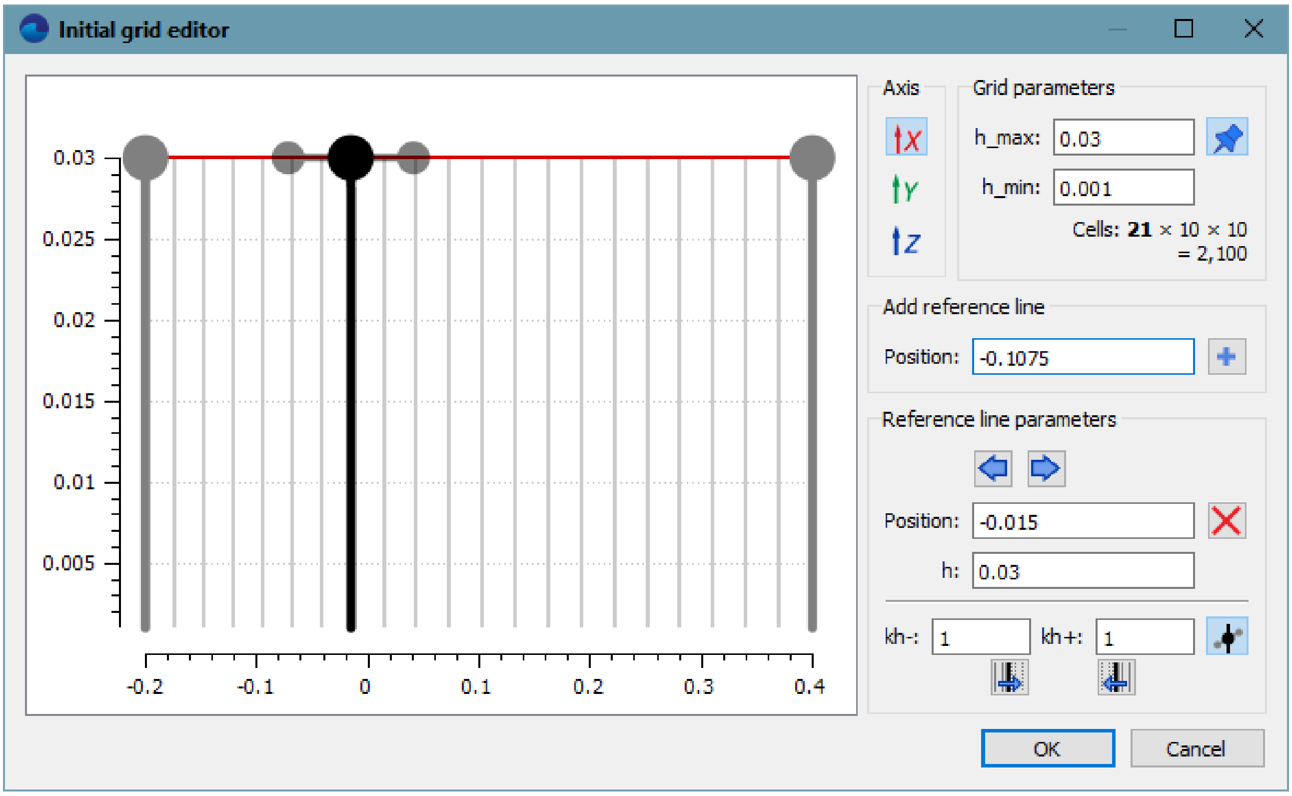
To specify in the **Initial grid editor** a non-uniform computational grid for OX, follow these steps:

- Specify the following values in the group of settings **Grid parameters** [1)]:

h_max = **0.03** [m]

h_min **= 0.001** [m]

- Insert the reference line with the coordinate **x = -0.015** [m]:
  - In the **Add reference line** group of settings, specify **Position = -0.015**.
  - Click the button  **(Add a reference line with the selected position)**. After this the new reference line will appear in the graphical pane of the **Initial grid editor** and in the **Position** field in the **Add reference line** group of settings the default value **-0.1075** will appear, which would be used for another reference line, in the case if it would be added.



- Specify parameters of reference lines [2)] :

- in the graphical pane of the **Initial grid editor** select the line with coordinate **x =-0.2** [m] (the leftmost line)
- in the group of settings **Reference line parameters** specify:

  h **= 0.03** [m]

  (don't change **h** *if it already has this value)*

- in the graphical pane of the **Initial grid editor** select the line with coordinate **x =-0.015** [m] (the middle line)
- in the group of settings **Reference line parameters** specify:

  h **= 0.001** [m]

  kh- **= 1**

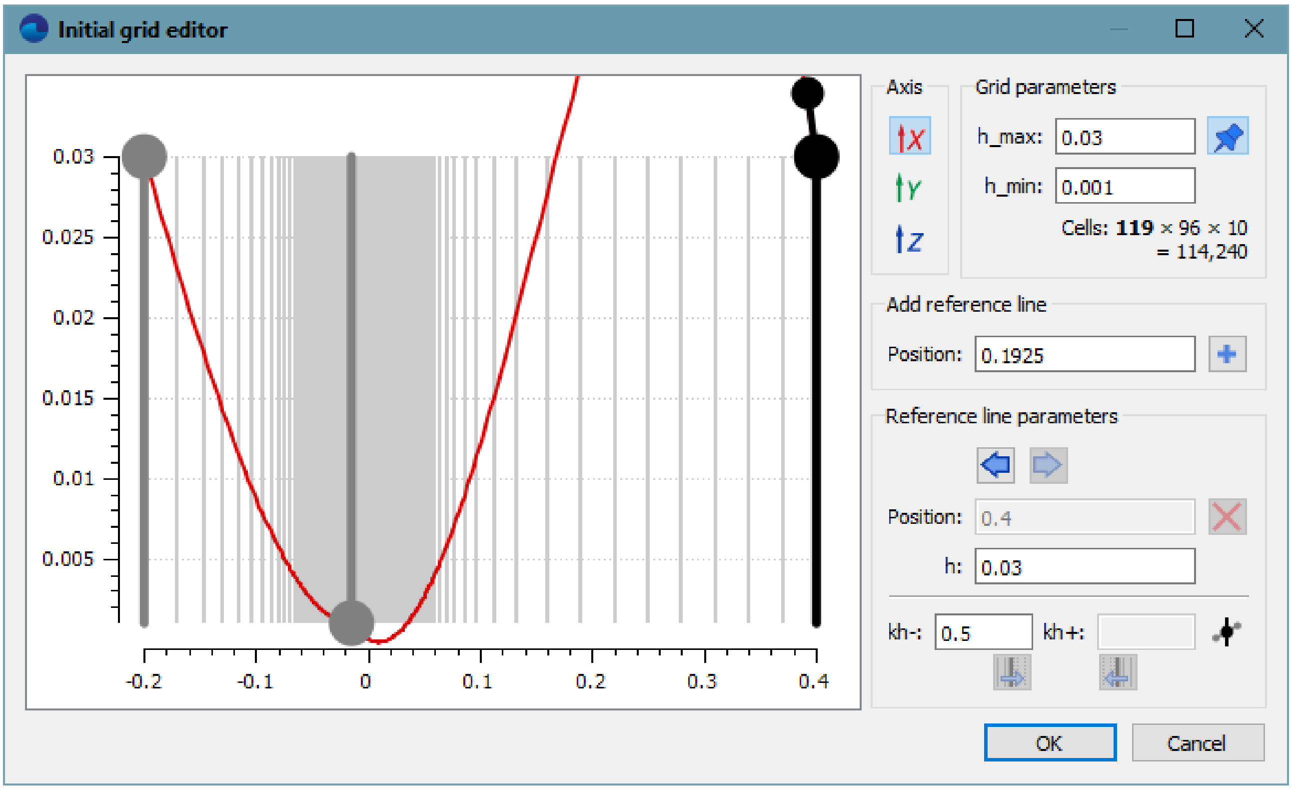
  kh+ **= 0.9**

- in the graphical pane of the **Initial grid editor** select the line with coordinate **x =0.4** [m] (the rightmost line)
- in the group of settings **Reference line parameters** specify:

  h **= 0.03** [m]

  (don't change **h** *if it already has this value)*

  kh- **= 0.5**



To specify in the **Initial grid editor** a non-uniform computational grid for OY, follow these steps:

- Click the button Y, to switch the editor to defining the grid across the **Y** axis.

Specify the following values in the group of settings **Grid parameters** [1]:

h_max = **0.03** [m]

h_min = **0.001** [m]

- Insert the reference line with the coordinate **y = 0** [m]:

○ set **Insert = 0**

○ click on the **Insert** button

  ○ In the **Add reference line** group of settings, specify **Position = 0**. *(don't change* **Position** *if it already has this value)*

  ○ Click the button **(Add a reference line with the selected position)**. After this the new reference line will appear in the graphical pane of the **Initial grid editor** and in the **Position** field in the **Add reference line** group of settings the default value **-0.1** will appear, which would be used for another reference line, in the case if it would be added.

- Specify parameters of reference lines [2] :

○ in the graphical pane of the **Initial grid editor** select the line with coordinate **y =-0.2** [m] (the leftmost line)

○ in the group of settings **Reference line parameters** specify:

h = **0.03** [m]

(don't change **h** *if it already has this value)*

kh+ = **1.5**

○ in the graphical pane of the **Initial grid editor** select the line with coordinate **y = 0** [m] (the middle line)

○ in the group of settings **Reference line parameters** specify:

h = **0.001** [m]

kh- = **1.2**

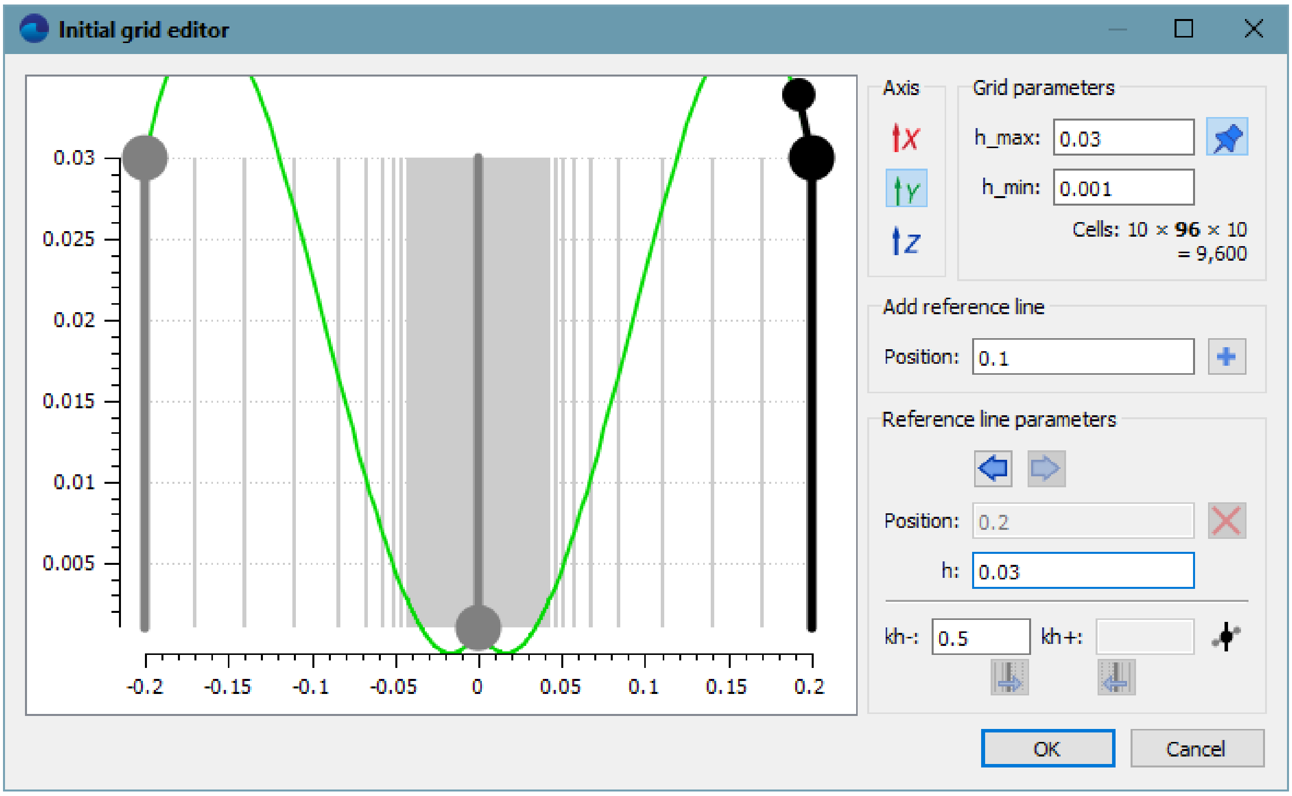
kh+ = 0.8

- in the graphical pane of the **Initial grid editor** select the line with coordinate **y = 0.2** [m] (the rightmost line)
- in the group of settings **Reference line parameters** specify:

h = 0.03 [m]

(don't change **h** *if it already has this value)*

kh- = 0.5



Then click the **OK** button.

Note:

[1)] Entering a value into a numerical field occurs when you press the **Enter** key on the keyboard, when the cursor locates in the field, or also when you relocate the cursor into another input field.

Transforming the 3D grid to a 2D grid

When you finish your work with the **Initial grid editor**, the program will create a uniform grid in the direction OZ consisting of 11 lines by default (with 10 cells). To create the required 2D setting, you have to specify the OZ direction as non-computational. To do so, in properties of the specify in **Initial grid** specify:

Grid structure = 2D

Plane = XY

In the **Properties** window of the **Initial grid** click **Apply**.

After this, a locked **nZ=1** value will be automatically set in properties of the **Initial grid**. For so long as **Grid structure = 2D** and **Plane = XY** are specified, the value of the **nZ** parameter is **1** and cannot be changed.

Specifying **Grid structure = 2D** and **Plane = XY** prevents applying of adaptation in the direction OZ (when the grid is 3D, i.e. when **Grid structure = 3D**, adaptation splits the cells to 8 portions, into halves along each direction OX, OY, OZ). When **Grid structure = 2D** and **Plane = XY**, the grid will become non-computational along axis OZ (it will always have only one cell along this axis).

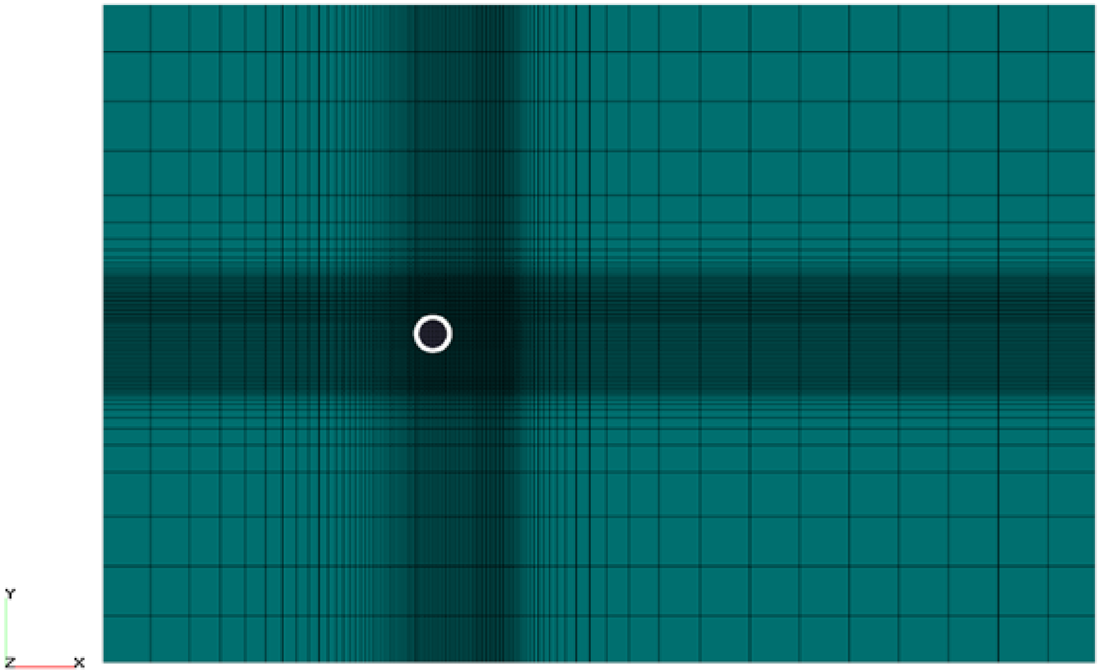
Specifying a non-computational direction blocks adaptation of cells along this direction so this allows to substantially economize the number of computational cells.

Visualization of the Initial grid

The resulting **Initial grid** is displayed:

- in **Preprocessor**, when the **Computational grid > Initial grid** element is selected in the project tree
- in **Postprocessor** by the **Initial grid** layer. The **Initial grid** layer is created automatically at creating the initial grid in **Preprocessor**.

To color the grid lines in black, specify **Lines > Color = Black** in properties of the **Initial grid** layer in **Postprocessor**.



4.1.1.5 Adaptation of the computational grid

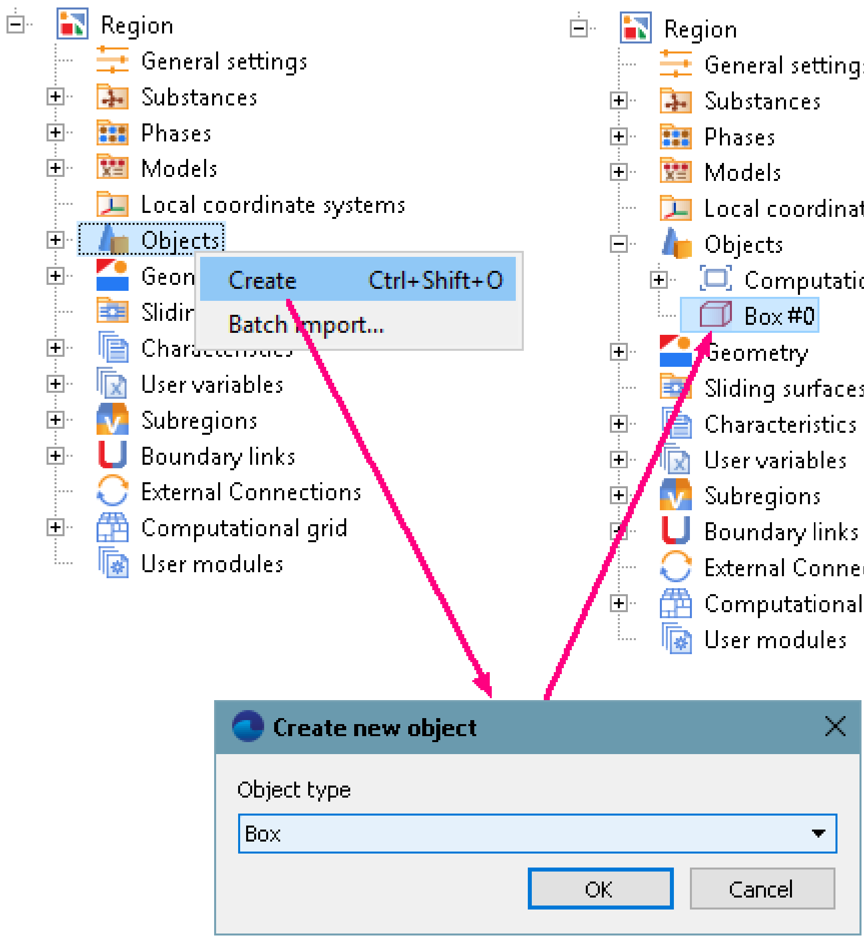
In this example, you have to solve the boundary layer around the cylinder and a vortex shedding zone behind the cylinder. For this it is necessary to make an adaptation of the grid in the volume of a box around the cylinder (see details in sections *"Splitting and merging cells of grid" and "Adaptation"* in the documentation of *FlowVision.*).

Specifying grid adaptation in an object consists of two steps:

1. Specifying the object of adaptation
2. Specifying the adaptation criteria

To specify the object of adaptation, follow these steps:

- In **Preprocessor** tab, in the context menu of the **Objects** folder, select **Create**
- In the **Create new object** window select **Object type** = **Box**



- In the **Properties** window of the **Box #0** specify:

Location

Reference point

X **= 0.01** [m]

Y **= 0** [m]

Z **= 0.005** [m]

Size

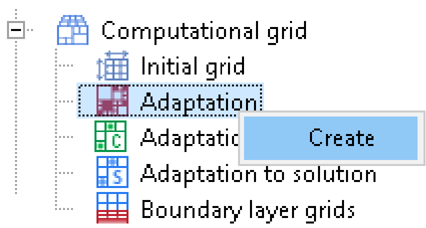
X **= 0.05** [m]

Y **= 0.04** [m]

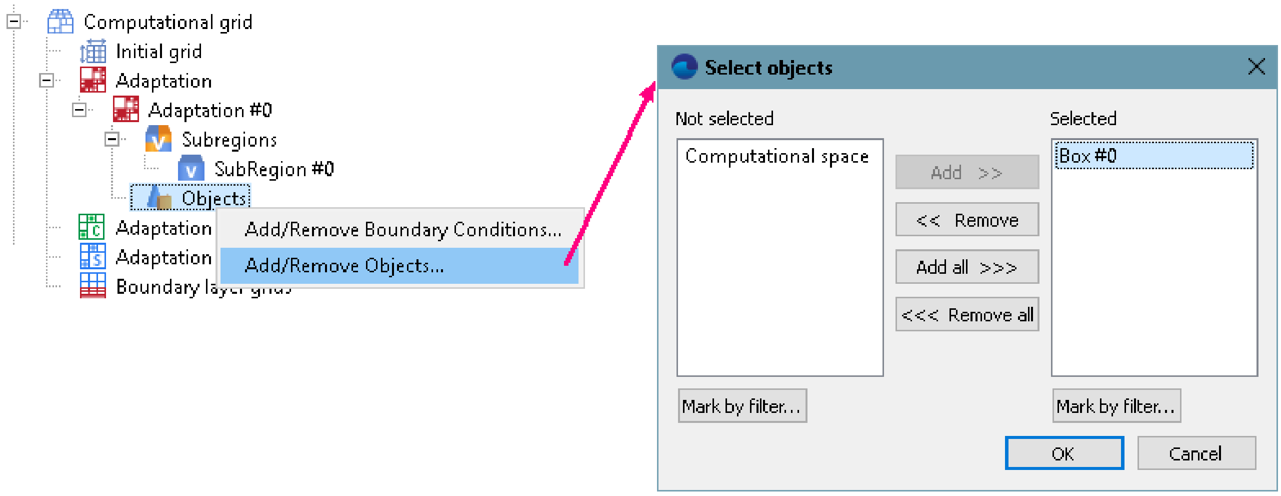
Z **= 0.01** [m]

Create the element **Adaptation #0** and specify its parameters:

- From the context menu of the folder **Computational grid > Adaptation**, select the **Create** command:



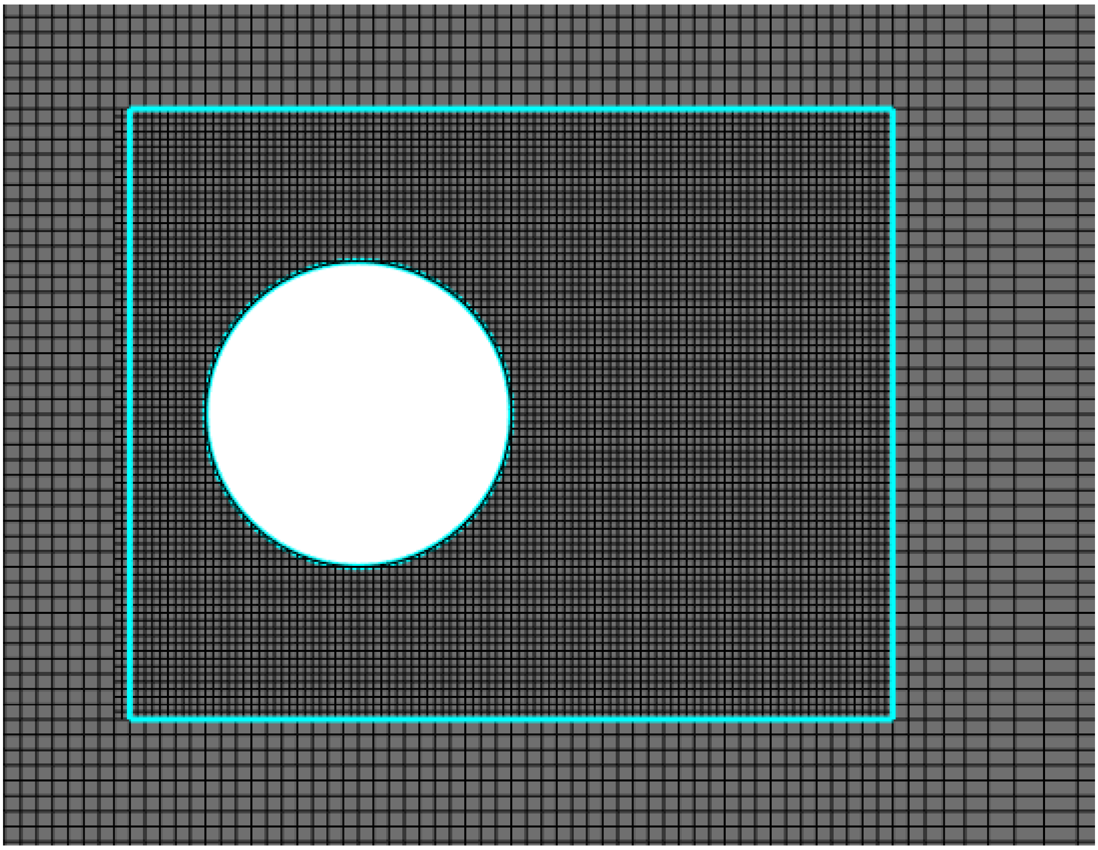
- From the context menu of the element **Computational grid > Adaptation > Adaptation #0 > Objects** select the command **Add/Remove Objects** and in the **Select objects** dialog box, which opens, place **Box #0** into the pane **Selected** and click **OK**:



- In the **Properties** window of the new created element **Adaptation #0** specify:

Enabled	**= Yes**
Max level N	**= 1**
Split/Merge	**= Split**
Layers	
Layers for Level N	**= 4**

Note, that in the **Properties** window of the element **Computational grid > Adaptation > Adaptation #0 > Objects > Box #0** it is set **Parts > Select = Volume**, which means adaptation within the volume of the geometry object **Box #0**.



> As in this exercise there is only one **Subregion**, you don't have to add **SubRegion #0** into the folder **Computational grid > Adaptation > Adaptation #0 > Subregions** using the **Add/Remove** command from the context menu; **SubRegion #0** is added to this folder automatically at creation of **Adaptation #0**.

In order to see the computational grid with adaptation, create a layer **Computational grid** on the **Computational space**. The layer is displayed after its computation is done.

4.1.1.6 Parameters of calculation

Specify the properties of the time step in the **Solver** tab of the **Project** window (**Solver > Time step**):

Method	**= In seconds**	
Constant step	**= 0.25**	[s]

4.1.1.7 Stopping conditions

In this example we recommend that you specify stopping condition based on the X-component of the force acting on the surface of the cylinder.

Create **Characteristics**:

- In the **Preprocessor** tab, create a **Supergroup** on the surface of the boundary condition **Wall** using the command **Create supergroup > In Preprocessor** from the context menu of the boundary condition's item in the project tree.
- Create **Characteristics** on the new just created **Supergroup on "Wall"**.
- In properties of **Characteristics #0** specify: **Variable > Variable = Pressure**.

Specify a **Stop criterion**:

- in the **Solver** tab, from the context menu of the folder **Stopping conditions > User values**, select **Create**.
- In the **Properties** window of the new just created **Stop criterion #0** specify:

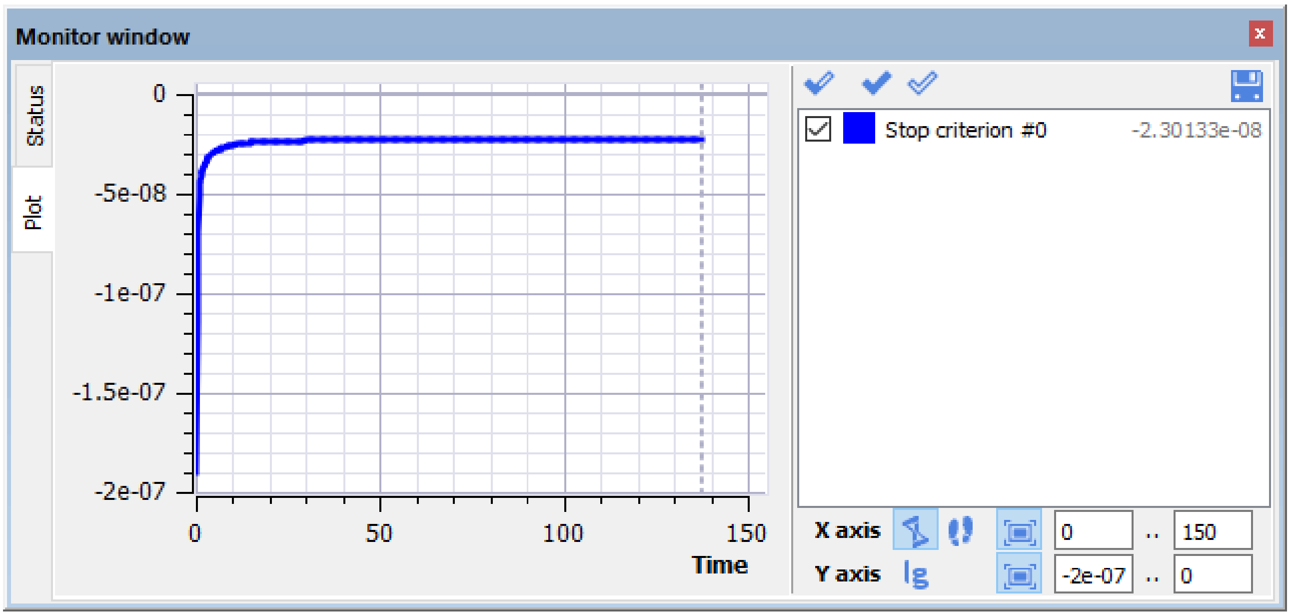
Level	**= 1e-6**
Object	**= Characteristics #0 (Supergroup on "Wall")**
Variable	**= F fluid**
Component	**= X**

4.1.1.8 Visualization

To view the dynamics of the solution during the computation, prior to computation, specify visualization of:

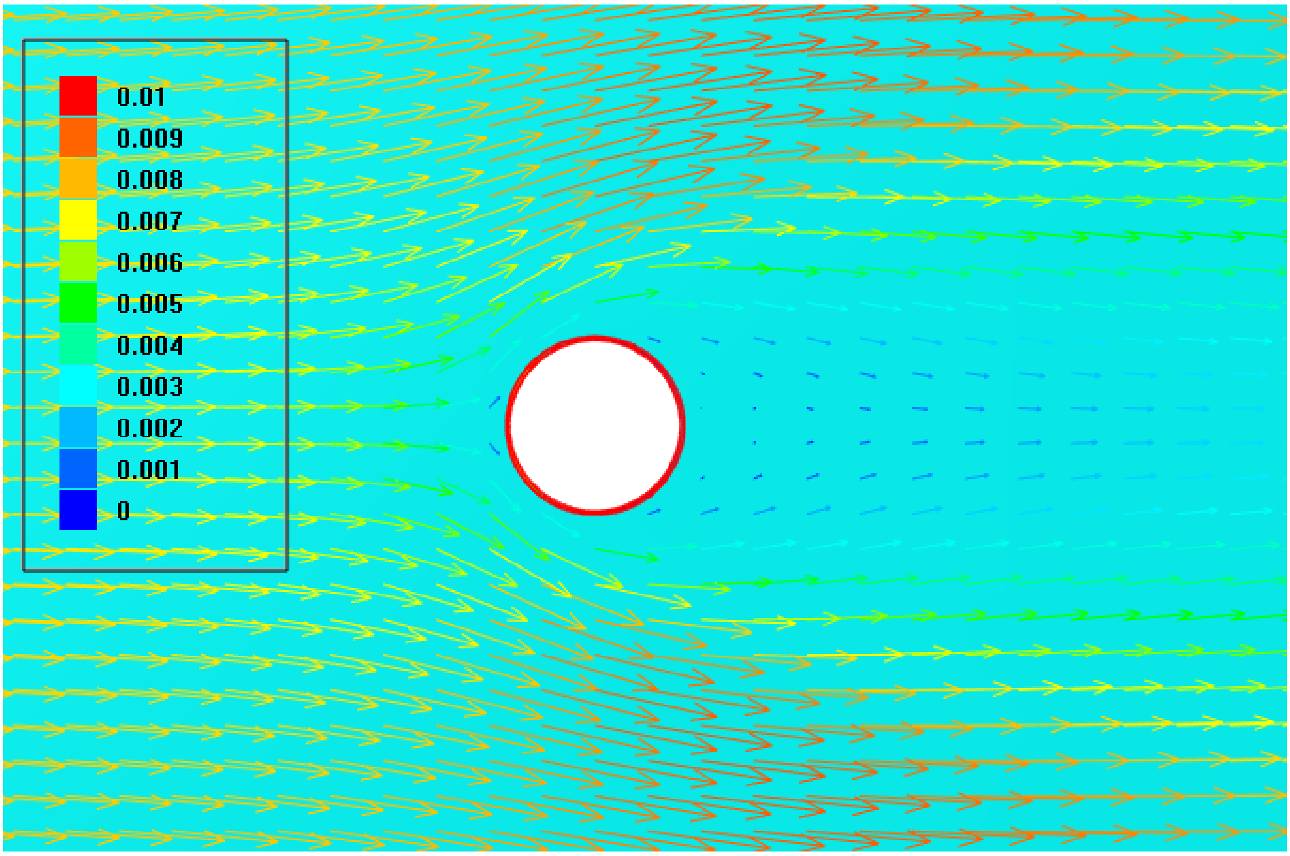
1. **The dynamics of the X-component of the force**, which acts on the cylinder's surface.
2. **Velocity distribution** in the plane of the flow.
3. **Pressure distribution** on the surface of the cylinder.

4.1.1.8.1 Force variation



View the dynamics of X-component of the force acting on the cylinder, on the **Plot** tab in the **Monitor** window.

4.1.1.8.2 Velocity distribution



- In the **Properties** window of **Plane #0** specify:

Object

Normal

X	**= 0**
Y	**= 0**
Z	**= 1**

(to direct a **Plane**'s normal along the axis **Z**, you can also click the **Operations >**  button in the **Properties** window of the **Plane**)

- On **Plane #0** create a **Vectors** layer and in its properties specify:

Grid

Size 1	**= 100**
Size 2	**= 100**

Coloring

Variable

Variable	**= Velocity**

Value range

Mode	**= Manual**
Max	**= 0.01**
Min	**= 0**

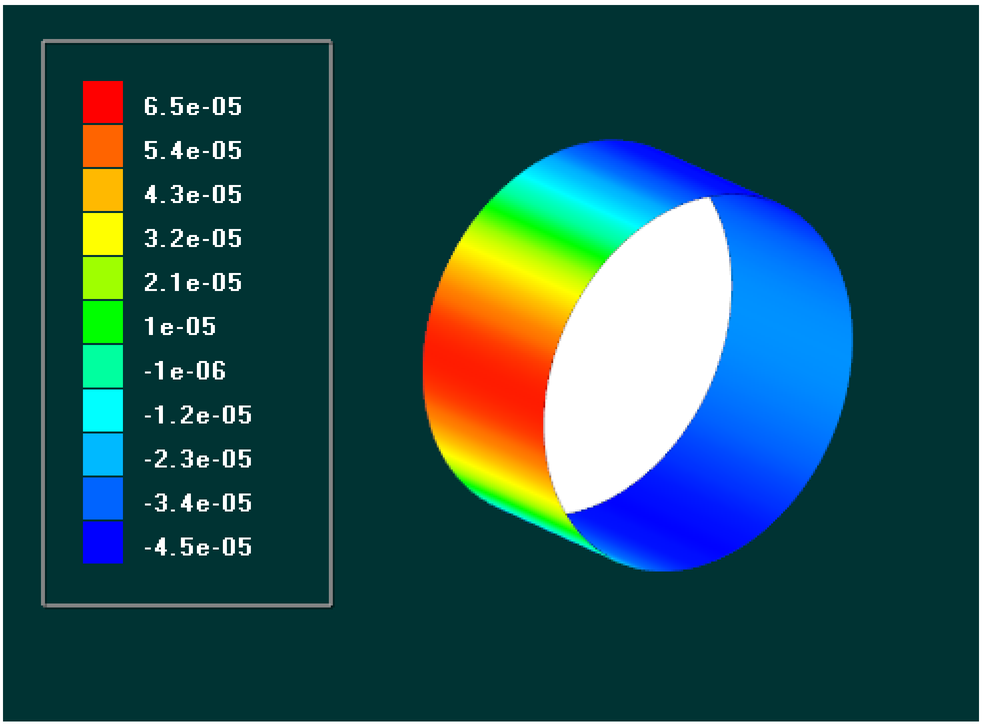
Palette

Appearance

Enabled	**= Yes**
Color	**= Black**

Note, that the program will automatically specify in the **Properties** window the variable, which is used to build the vectors, **Variable > Variable = Velocity**.

4.1.1.8.3 Pressure distribution



- Create a layer **Color contours** on **Supergroup on "Wall"**.
- In the **Properties** window of the new **Color contours** layer, specify:

Variable

Variable	**= Pressure**

Shift	**= 0.0001**

Value range

Mode	**= Manual**
Max	**= 6.5e-5**
Min	**= -4.5e-5**

Palette

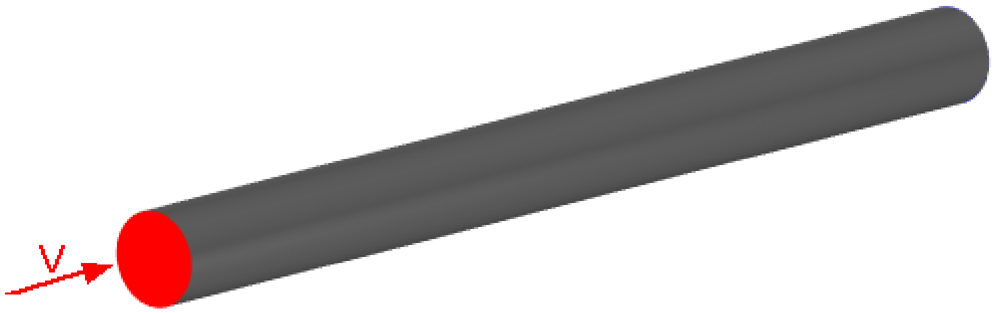
Appearance

Enabled = **Yes**

Note: If you wish to hide some layers or objects displayed in the **View** window, use the **Hide** command from their context menus.

4.1.2 Time-varying flow in a tube

Axi-symmetric laminar water flow with variable inlet velocity is considered in the given example.



Dimensions:

Length of the tube	L	= 0.5	[m]
Diameter of the tube	D	= 0.04	[m]

Inflow parameters:

Velocity on inlet: $V_{inl} = 0.005 \cdot \left(2 + \sin\left(\frac{2\pi t}{1000}\right)\right)$ [m s^{-1}]

Fluid parameters:

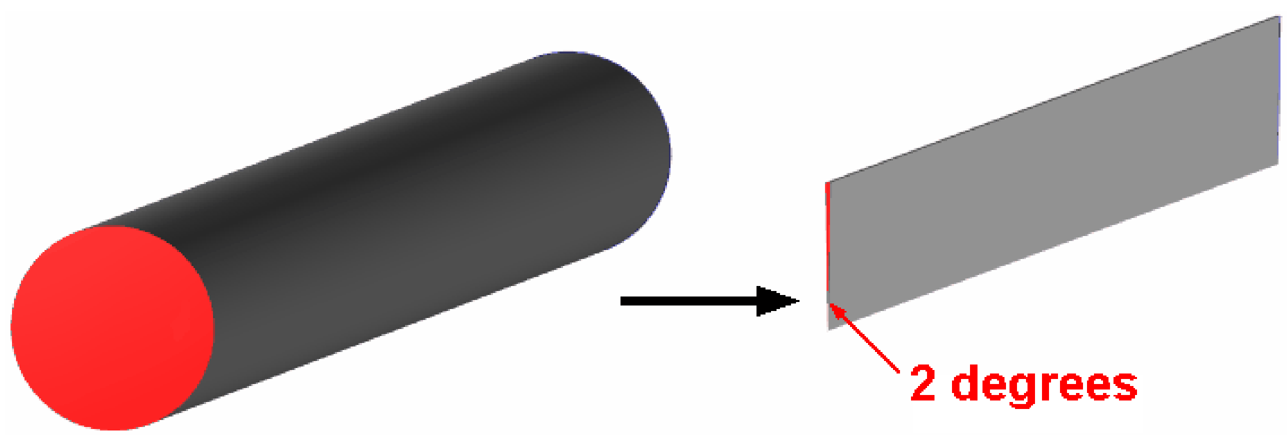
Density	ρ	= 1000	[kg m^{-3}]
Viscosity	μ	= 0.001	[kg m^{-1} s^{-1}]

Reynolds number: $Re = \frac{\overline{V_{inl}} D\rho}{\mu} = \frac{0.01 \cdot 0.04 \cdot 1000}{0.001} = 400$

Geometry: a sector of a cylinder that will be created by means of *FlowViwion*

Project: `Tube_VarMassFlow`

4.1.2.1 Computational domain



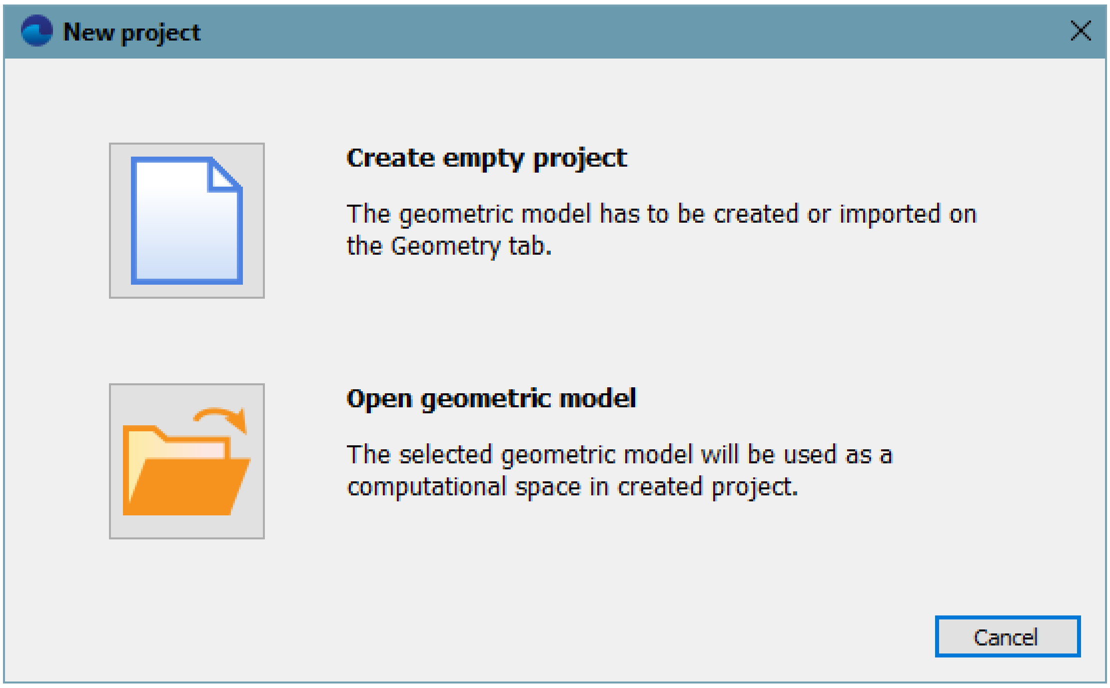
In order to simulate the axisymmetric flow you should create a computational domain consisting of a sector with a small opening angle (e.g., 1-2 degrees)*).

There are the following methods of specifying the computational domain in *FlowVision*:

- Loading parametric geometry models created in external CAD software (this method is used in previous exercises)
- Creating a geometry model immediately in *FlowVision* (this method is used in this exercises) either in the **Preprocessor** tab (as described below in this exercise) or in the **Geometry** tab (as described in the Natural convection exercise).

Create an empty *FlowVision* project using the **File > Create** command from the main menu.

The **New project** dialog box will open:



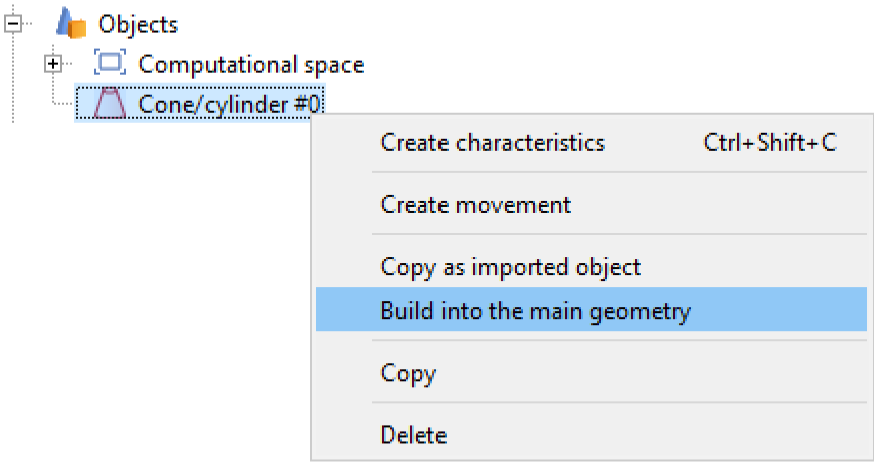
In this dialog box click the  (**Create empty project**) button. An empty project will be created with the **Geometry** tab opened.

To create the computational domain, follow these steps in the project tree in the **Preprocessor** tab:

- In the **Objects** folder create **Cone/cylinder #0** (select **Create** from the context menu of the folder **Objects** and in the **Create new object** dialog box set **Object type** = **Cone/cylinder**).
- In properties of **Cone/cylinder #0** specify:

Parameters		
Height	**= 0.5**	[m]
Radius 1	**= 0.02**	[m]
Radius 2	**= 0.02**	[m]
Base ratio	**= 1**	
Sector		
Arc start	**= -1**	[degree]
Arc angle	**= 2**	[degree]
Approximation		
Subdivisions	**= 9**	
Per sector	**= Yes**	

- Click **Apply** in the **Properties** window of **Cone/cylinder #0**.
- In the project tree open the context menu of **Cone/cylinder #0** and select there the command **Build into the main geometry**:



After this **SubRegion #0** will appear in the folder **Subregions**. Geometry of **SubRegion #0** presents a sector of the tube with opening angle of 2 degrees and symmetrical relating to the plane XY (1 degree to the left and 1 degree to the right from the plane XY).

Notes:

*) Further, when specifying the project you have to:

1. When you specify Boundary conditions on the surfaces of the planes of symmetry, specify boundary conditions **Symmetry**.
2. When you form the Initial grid, specify a non-computational direction as **Z**.

The computational domain can also be set in the **Geometry** tab, see description in the exercise Natural convection, section Creating the computational domain in the "Geometry" tab.

4.1.2.2 Physical model

Do the following steps in the tree of the **Preprocessor** tab:

In the folder **Substances**:

- Create **Substance #0**
- Specify the following properties of **Substance #0**:

Aggregative state	**= Liquid**	
Molar mass		
Value	**=0.018**	[kg mole^{-1}]
Density		
Value	**= 1000**	[kg m^{-3}]
Viscosity		
Value	**= 0.001**	[kg m^{-1} s^{-1}]
Specific heat		
Value	**= 4217**	[J kg^{-1} K^{-1}]

In the folder **Phases**:

- Create a continuous **Phase #0**.
- Add **Substance #0** into the folder **Phase #0 > Substances**.
- Specify in properties of the folder **Phase #0 > Physical processes**:

Motion **= Navier-Stokes model**

In the folder **Models**:

- Create **Model #0**

- Add **Phase #0** into the folder **Model #0 > Phases**
- Specify in the folder **Model #0 > Init. data > Init. data #0**:

Velocity (Phase #0)

X	**= 0.01**	[m s^{-1}]

4.1.2.3 Boundary conditions

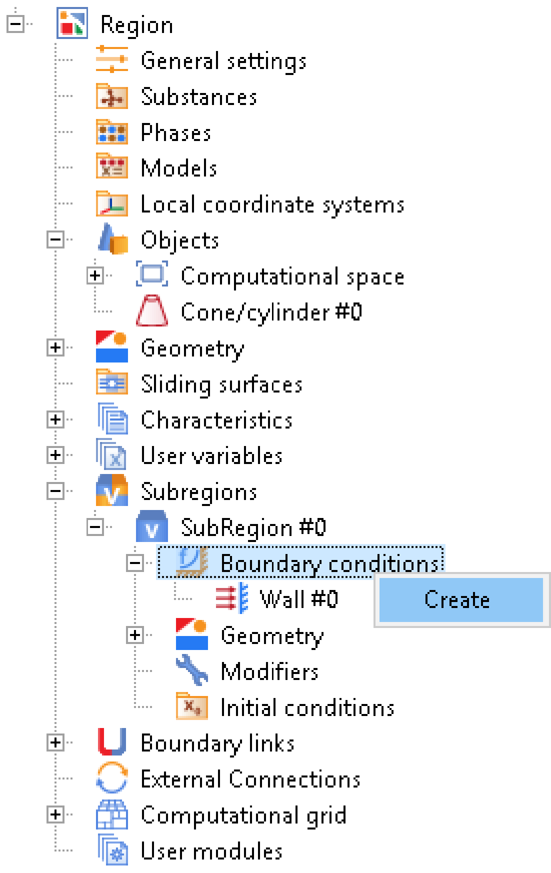
In the **Properties** window of the **SubRegion #0** specify:

Model = Model #0

When after your import of geometry the amount or arrangement of boundary conditions differ from those desired, you should to specify the boundary conditions manually. Definition of boundary conditions consists of 3 steps:

1. Create boundary conditions
2. Place the boundary conditions
3. Specify parameters of the boundary conditions

To create a new boundary condition, select **Create** in the context menu of the **Boundary conditions** folder:



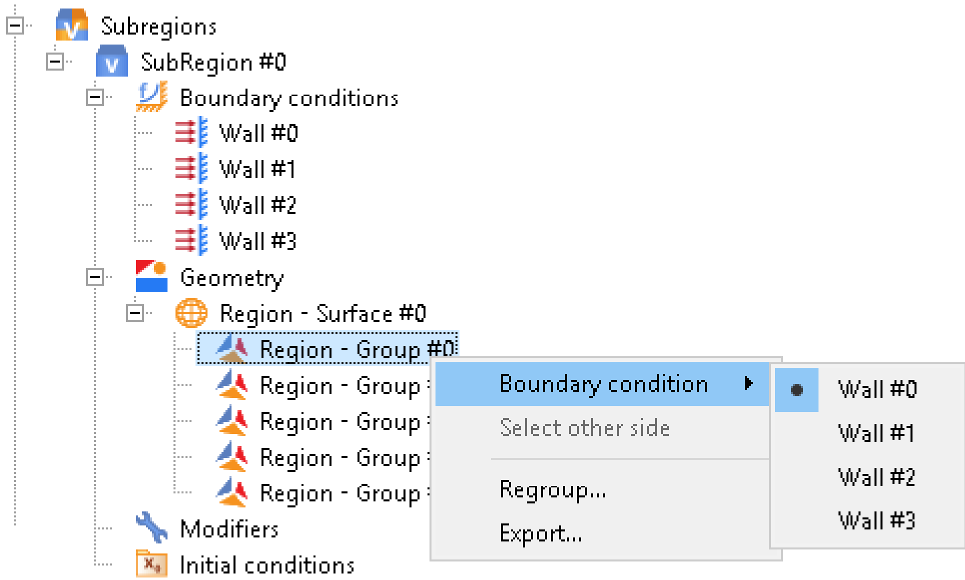
The **Boundary conditions** are placed on **Groups**. There are two ways of placing the **Boundary conditions**:

1. *either* in the **View** window
2. *or* in the **Geometry** folder

In order to place a **Boundary condition** on a **Group** in the **View** window do the following:

- switch to the **Selection mode** by clicking on button 

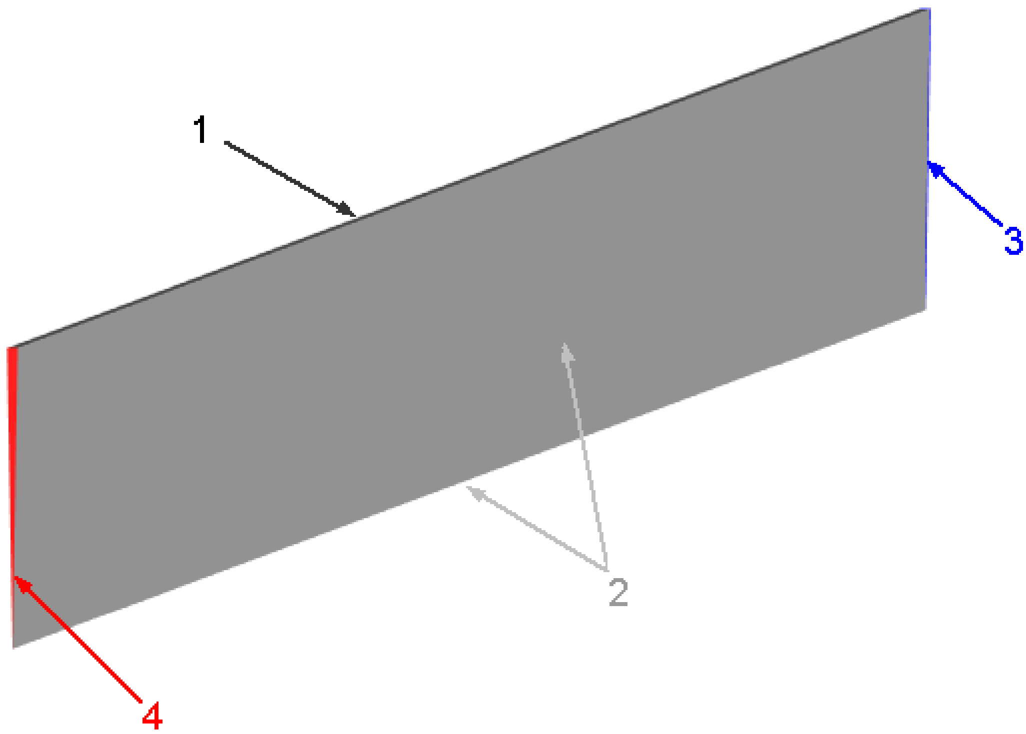
- select the appropriate **Group** in the **View** window
- in its context menu select a **Boundary condition >** ***B. Cond. #N***



In order to place a **Boundary condition** on a **Group** in the **Geometry** folder do the following steps:

- apply the menu command **File > Preferences** and specify:
  Display
  Show all groups **= Yes**
- in the **Properties** of the respective **Group** in the folder **Geometry > Region - Surface #0** specify:
  Boundary condition **=B. Cond. #i**

In this project, set the following boundary conditions:



Boundary 1

Name **= Wall**

Type = **Wall**
Variables
Velocity (Phase #0) = **No slip**

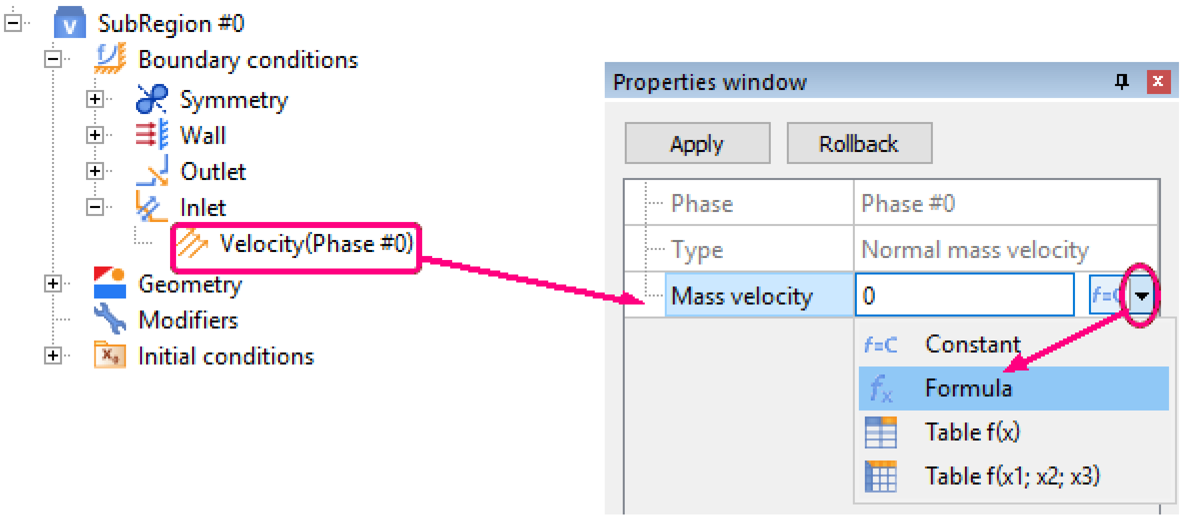
Boundary 2
Name = **Symmetry**
Type = **Symmetry**
Variables
Velocity (Phase #0) = **Slip**

Boundary 3
Name = **Outlet**
Type = **Free Outlet**
Variables
Velocity (Phase #0) = **Pressure**
Value = **0** [Pa]

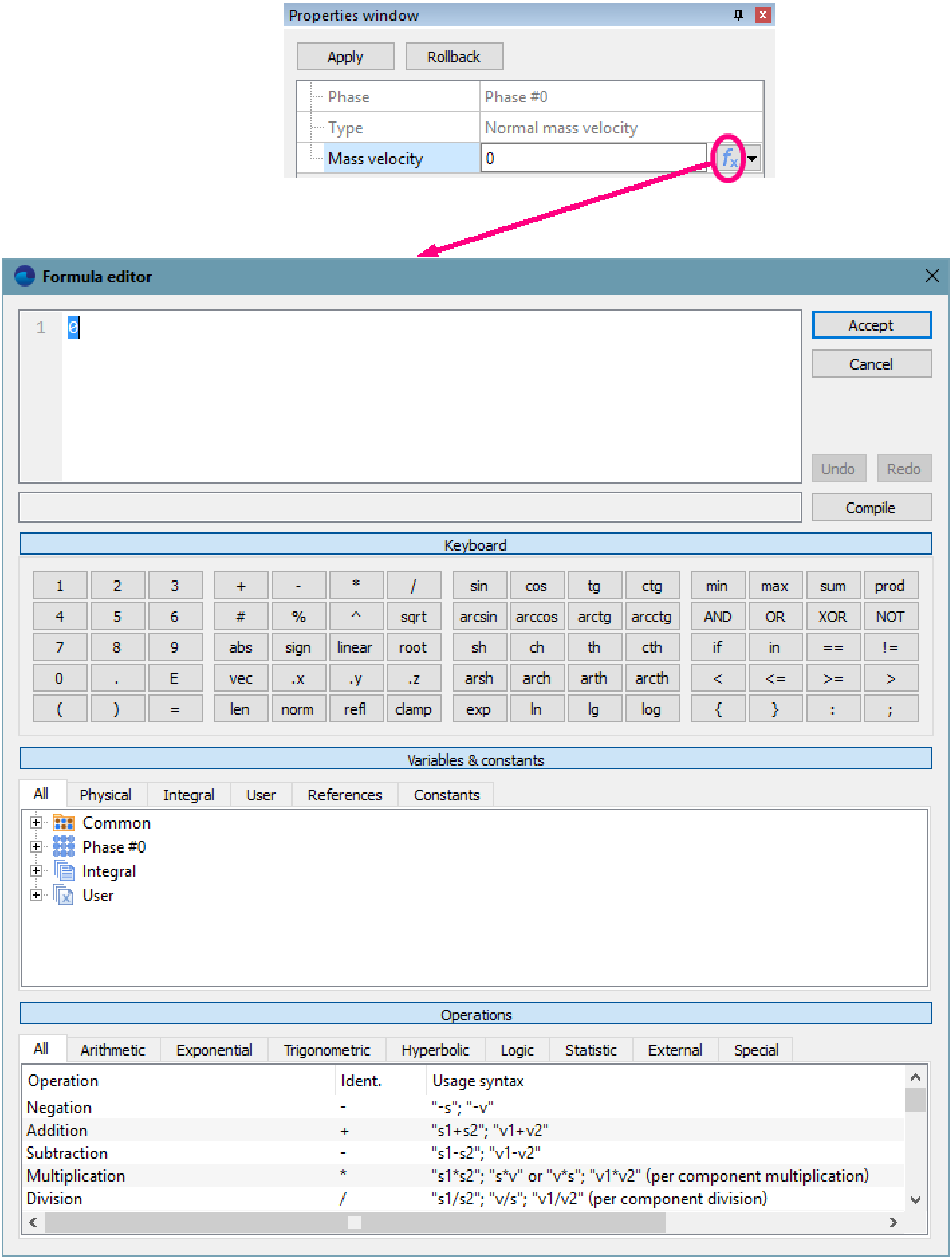
Boundary 4
Name = **Inlet**
Type = **Inlet/Outlet**
Variables
Velocity (Phase #0) = **Normal mass velocity**

To set the **Velocity** variable on the **Inlet** do the following steps:

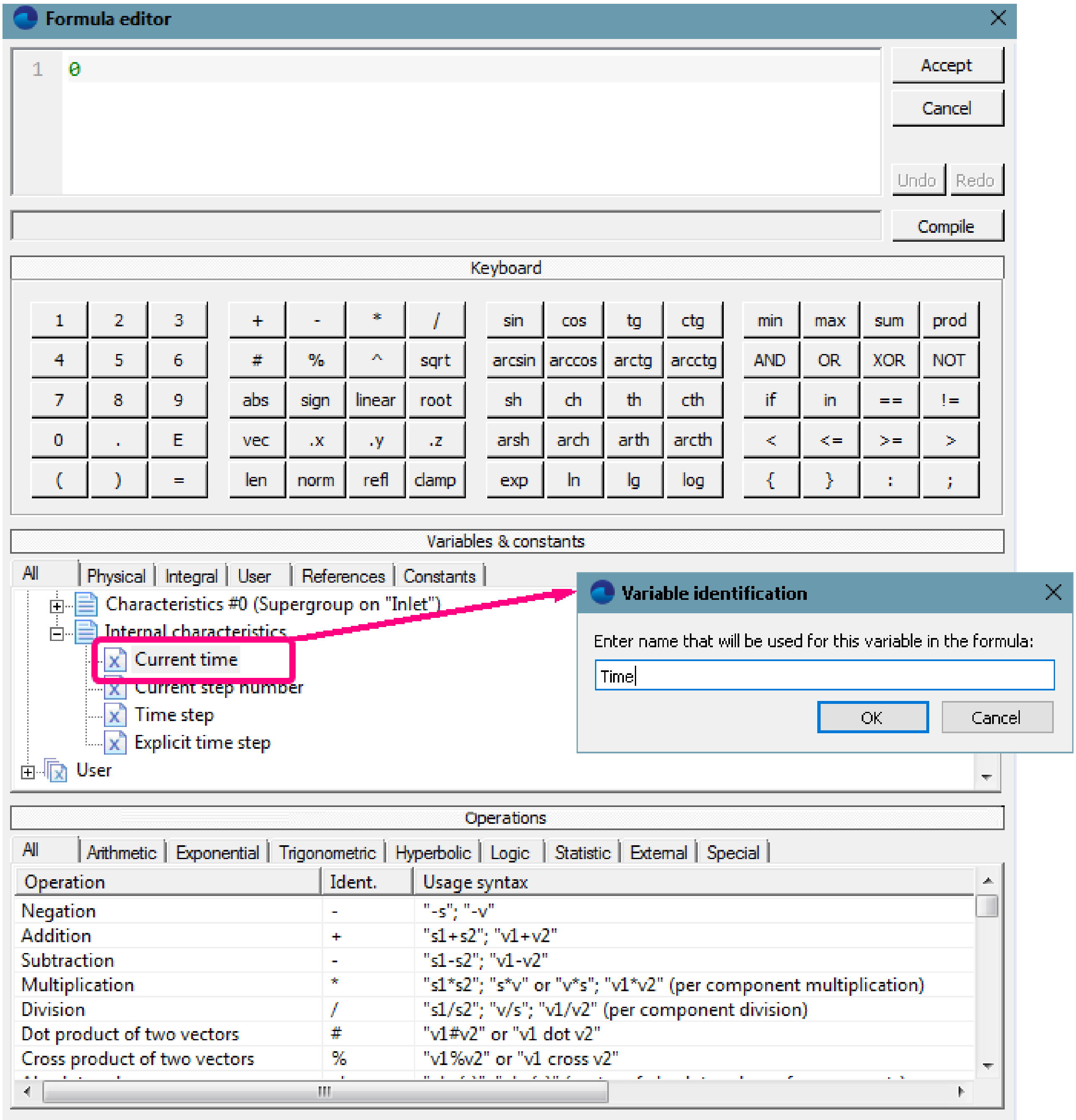
- In the **Properties** window of the boundary conditions for **Velocity** on **Inlet** change the type of entered value from **Constant** to **Formula**:



- Click on the button , which opens the **Formula editor**:

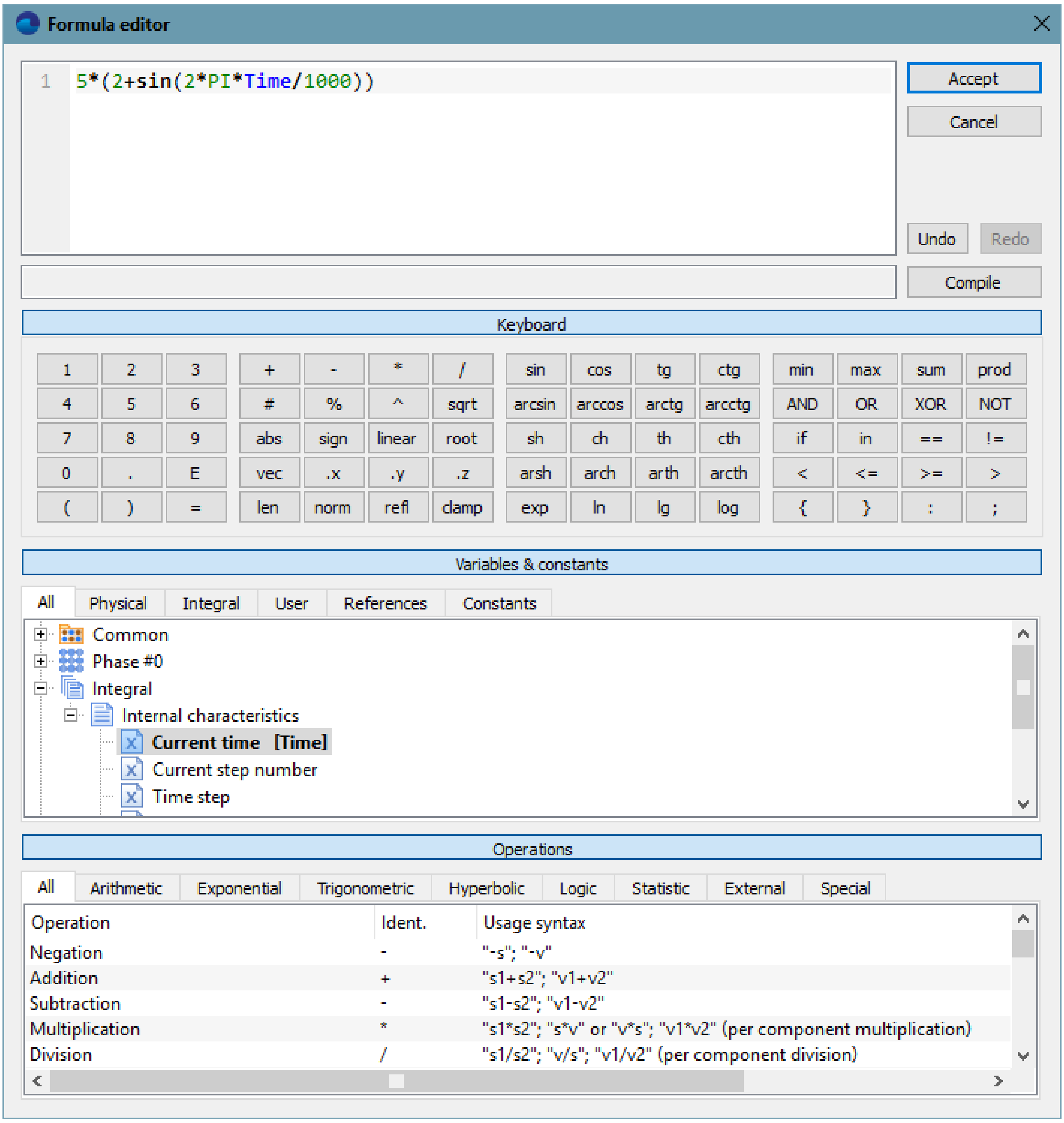


- Make an identification of the variable **Time**:
  - In group of settings **Variables & constants** (on the tab **All**) expand item **Integral > Internal characteristics > Current time** and double click on it.
  - In the **Variable identification** window that appears, specify the name of the variable as "**Time**".



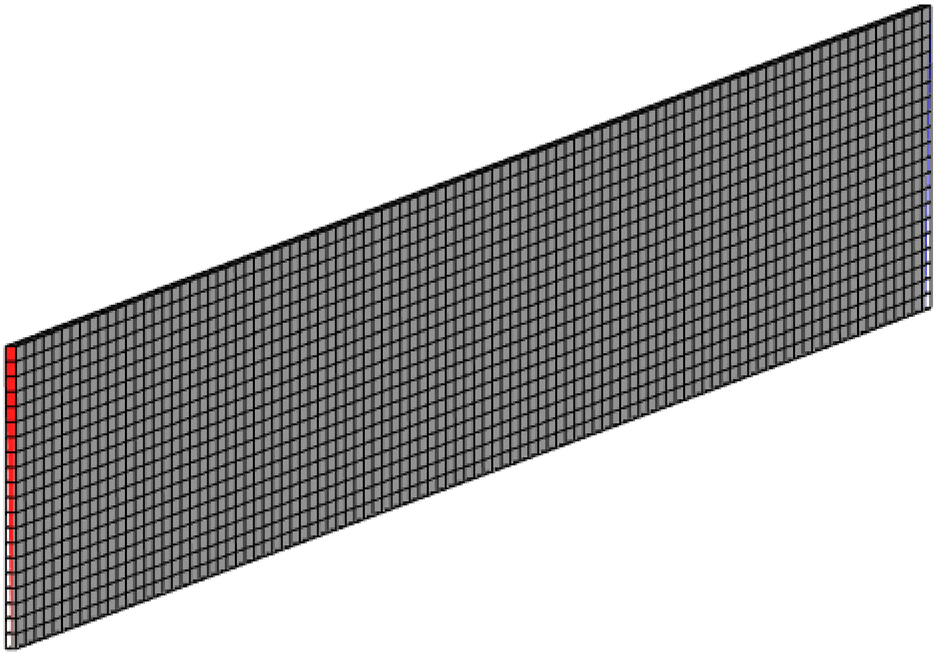
- Make an identification of the constant **PI**:
    - In group of settings **Variables & constants** (on the tab **Constants**) select **Pi number** and double click on it.
    - In the **Constant identification** window that appears, specify the name of the variable as "**PI**".
- In the **Formula editor**, specify the formula in the **Formula pane**:

$$5*\left(2+\sin\left(\frac{2*\mathrm{PI}*\mathrm{Time}}{1000}\right)\right)$$



- In the **Formula editor** click **Accept**.
- In the **Properties** window of the boundary conditions for **Velocity** on **Inlet** click **Apply**.

4.1.2.4 Initial grid



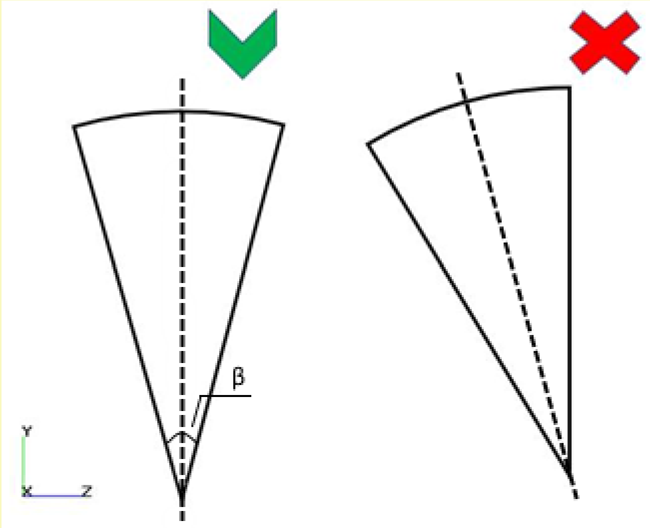
Specify in the **Properties** window of the **Initial grid**:

Grid structure	**= 2D**
Plane	**= XY**
nX	**= 100**
nY	**= 20**

In the **Properties** window of the **Initial grid** click **Apply**.

To use a sectoral axially symmetric geometry model in 2D setting, the model is to meet the following requirements:

- The geometry model is to be symmetrical relating to the plane that is perpendicular to the non-computational direction.



- The sector's angle β is to be ≤10 degrees.
- On the faces, which are oriented across the non-computational direction, boundary conditions of types **Symmetry** or **Connected** are to be set.

4.1.2.5 Parameters of calculation

In the **Solver** tab, in properties of the **Time step** element, specify:

Method	**= In seconds**	
Constant step	**= 10**	[s]

If you specified the non-computational direction not by using parameter **Grid structure** (or parameter **2D direction** in versions *FlowVision 3.12.02 - 3.12.04*) but by specifying the number of cells as **1** along the non-computational axis instead (as it was practiced in *FlowVision 3.11.01* and earlier versions), you should have to use the relative criterion of small cells (specify **Small Cells > Criterion = Relative** in properties of the element **Limiters > Limiters for calculation > Phase Limiters > Phase #N** in the project tree, in the **Solver** tab).

In versions *FlowVision 3.11.02* and higher, when a non-computational direction is set for simulations in narrow sectors, you should not change the default (**Absolute**) criterion of small cells.

4.1.2.6 Stopping conditions

You can use **Stop criteria** to visualize variation of a variable:

- in the Plot tab of the Monitor window
- *and/or* in the textual title in the **View** window (this is set in the **Postprocessor** tab, in properties of the root folder **3D-scene** by the **Title > User values > [N] > User stopper** parameter).

In cases when a **Stop criterion** is used for visualization only, you should not change the default zero value of its **Level** parameter.

A **Stop criterion**, for which **Level=0** is set, will never come into action and will not stop the project's computation but nevertheless it can be used for visualization.

In this exercise we will create **Stop criteria** to visualize dynamics of:

- Mass flow rate at the outlet (at the current step)
- Total mass flow at the outlet (for all the time of the computation)

Mass flow rate at the outlet at the current step

Let's start from creating **Characteristics** on the boundary conditions **Outlet**:

- In the project tree, in the **Preprocessor** tab, right-click the **Subregions > SubRegion #0 > Boundary conditions > Outlet** element.
- From the context menu that opens, select **Create supergroup > In Preprocessor**. A new object, **Supergroup on "Outlet"** will be created in the **Objects** folder.
- Open the context menu of **Supergroup on "Outlet"** and select there the **Create characteristics** command there. Create a new block of characteristics by the **Velocity** variable.

Create a **Stop criterion**:

- In the **Solver** tab open the context menu of the folder **Stopping conditions > User values** and select the **Create** command. A new element, **Stop criterion #0** will appear in this folder.
- In properties of the just created **Stop criterion #0** specify:

Object	**= Characteristics #0 (Supergroup on "Outlet")**
Variable	**= Mass flow-**

Sign in the name of the **Mass flow** variable (either "**-**" or "**+**") is specified based on direction of the normal to the surface of the **Outlet** boundary condition. The positive direction of the normal is directed inside the computational domain, so the mass flow that flows outside from the computational domain corresponds to the **Mass flow-** variable.

Total mass flow at the outlet (for all the time of the computation)

To provide time integrating of the mass flow on the outlet we will create two global **User variables**.

One of these **User variables** will store the total mass flow from the previous time step, and the other **User variable** will be used to calculate the cumulative mass flow.

$$Q_i = prev(Q) + q_i \tau_i = Q_{i\text{-}1} + q_i \tau_i$$

where:

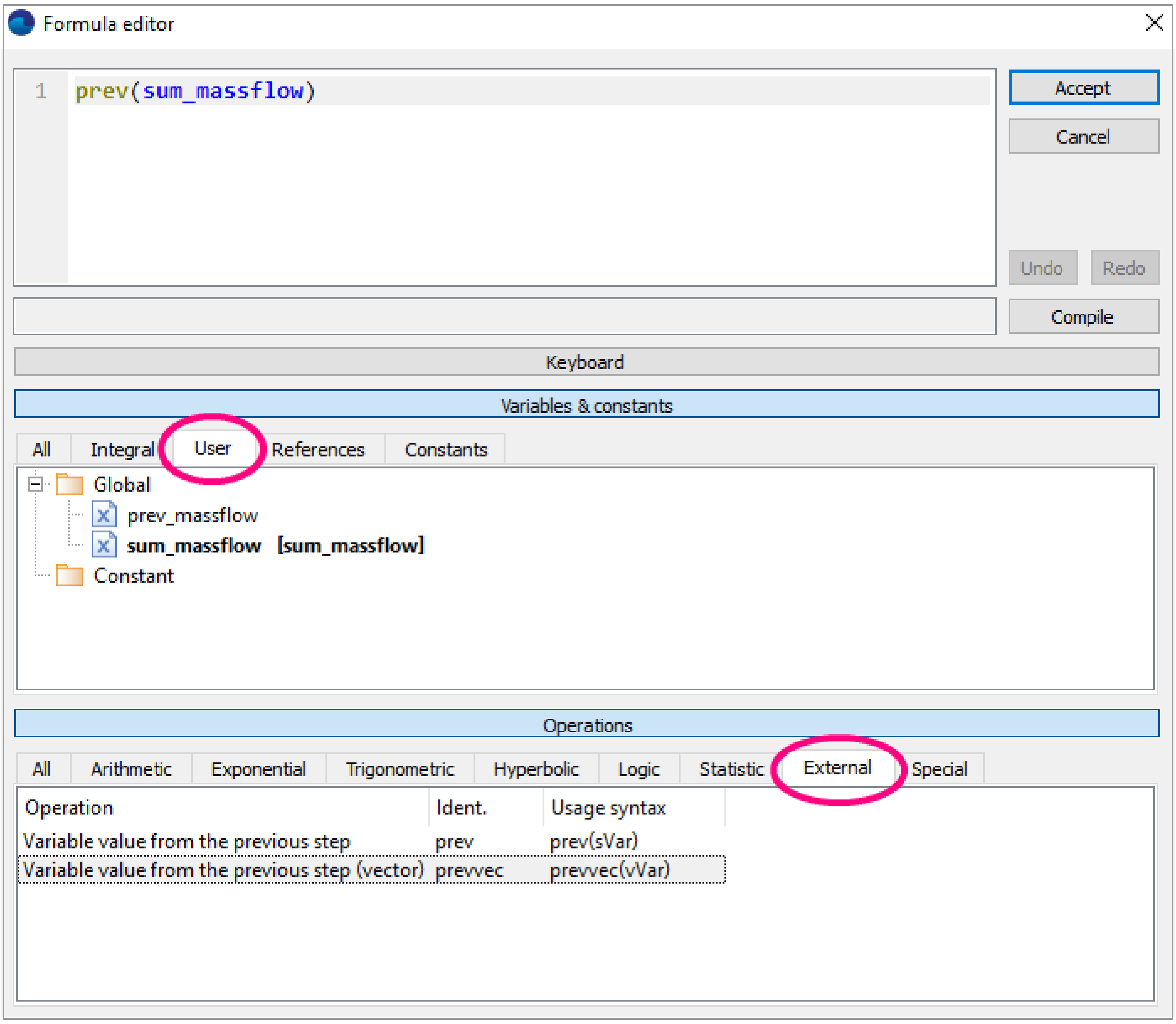
q_i is the mass flow, [kg/s], that flows outside the tube at the computational step i

τ_i is duration, [s], of the time step at the computational step i

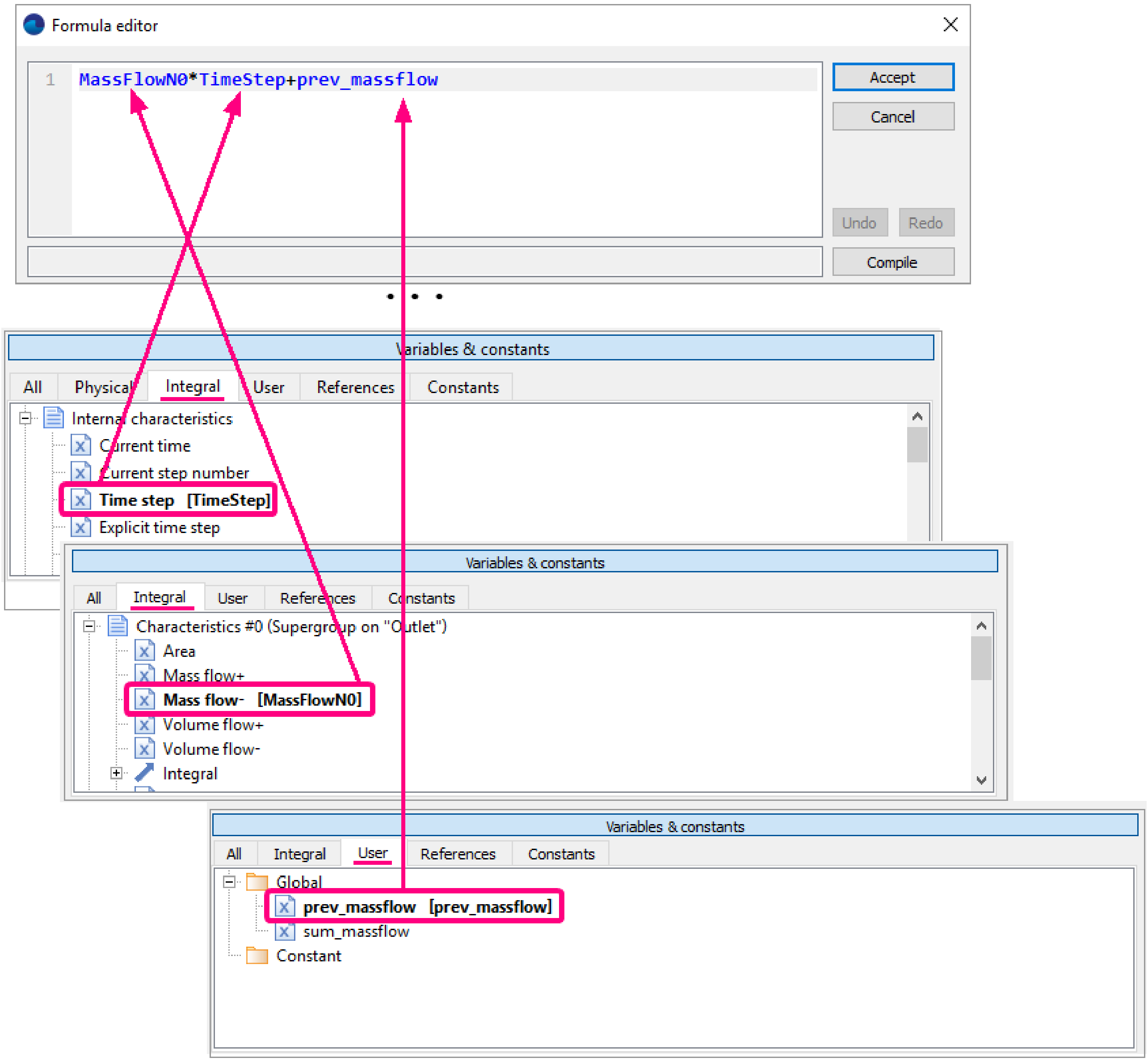
Q_i is the integrated (cumulative) mass flow, [kg], at the computational step i

In the **Preprocessor** tab:

- Open the context menu of the folder **User variables > Global** and select the **Create > Scalar** command. A new scalar variable **UGV #0** will appear in the folder **User variables > Global**.
- Similarly create another global scalar value, **UGV #1**.
- Rename the variables **UGV #0** and **UGV #1** as **prev_massflow** and **sum_massflow**.
- Specify values of the variables **prev_massflow** and **sum_massflow** in the **Formula editor**:
  - The variable **prev_massflow** specify by the formula `prev(sum_massflow)`.
  - The variable **sum_massflow** specify by the formula `MassFlowN0*TimeStep+prev_massflow`.



Specifying the formula for **prev_massflow**



Specifying the formula for **sum_massflow**

Create **Stop criterion #1** by selecting the **Create** command from the context menu of the folder **Stopping conditions > User values** and in properties of the **Stop criterion** specify:

Object = **sum_massflow**

4.1.2.7 Visualization

Viewing dynamics of the solution during the computation is described in sections below:

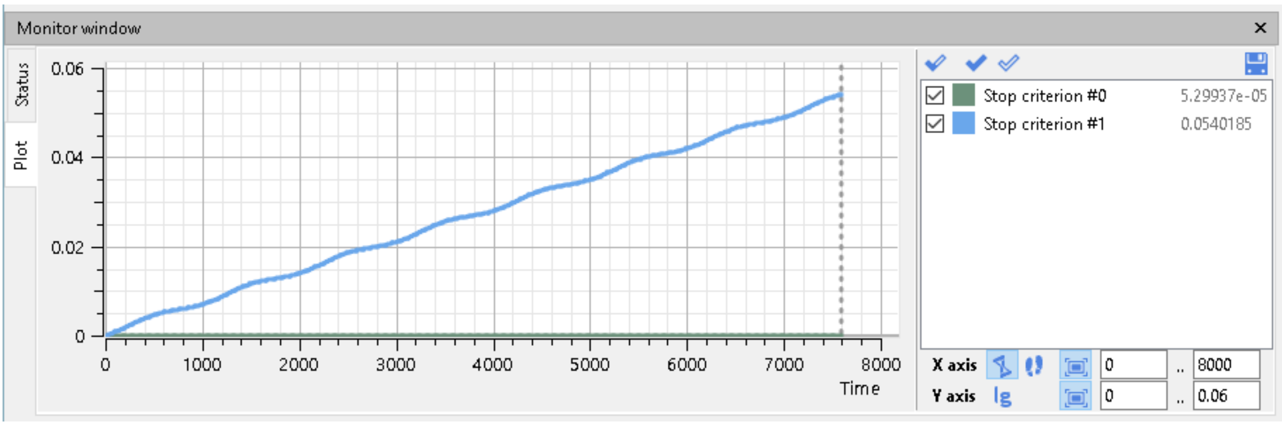
- Plots of mass flow
- Displaying text in the View window
- Velocity distribution in the plane of the flow.

4.1.2.7.1 Plots of mass flow

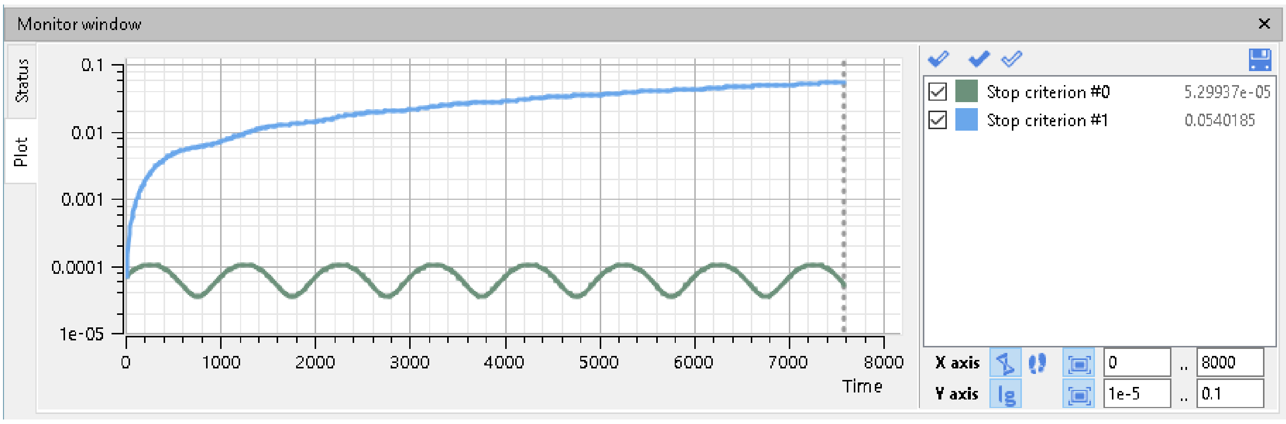
When the project is running, you can watch, in the **Plot** tab in the **Monitor** window, plots of **Stop criteria** that were created before:

- **Stop criterion #0** corresponds to the mass flow rate at the outlet at the current step.
- **Stop criterion #1** corresponds to the total mass flow at the outlet calculated cumulatively for all the time of the computation.

When the  screen button is pressed, these plots will be displayed in logarithmic scale for ordinates.



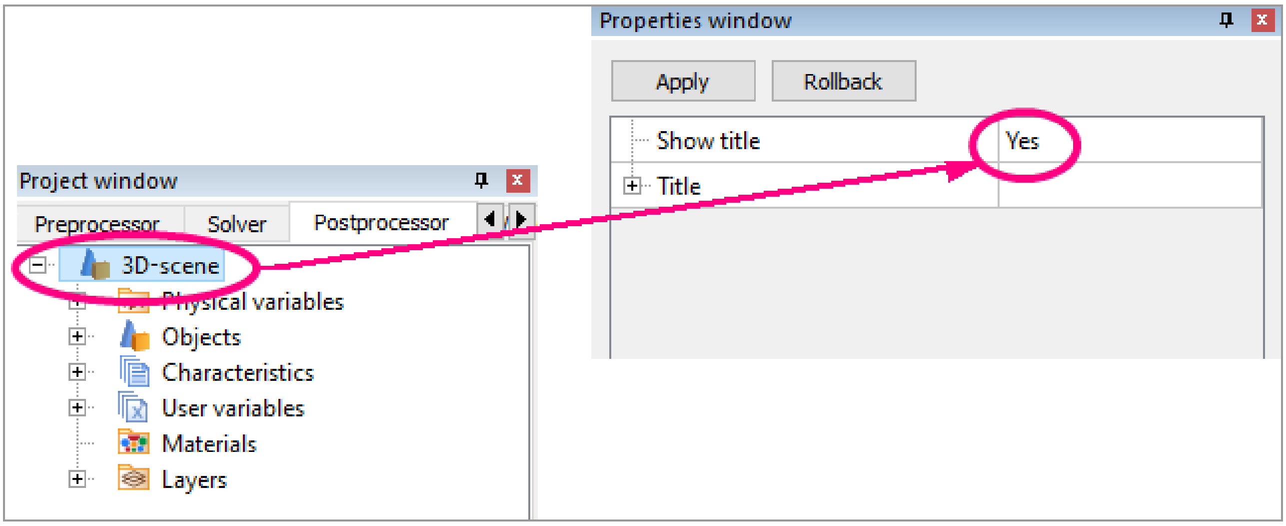
Plots of dynamics of **Stop criterion #0** and **Stop criterion #1** in the **Monitor** window



Plots of the same **Stop criteria** displayed in logarithmic scale for ordinates (the lg button is pressed)

4.1.2.7.2 Displaying text in the View window

In dynamic simulations, to display the calculated results as texts in the **View** window in the real-time mode, you can use group of parameters **Title > ...** in properties of the root element **3D-scene** in the **Postprocessor** tab of the project tree.



In properties of the root element **3D-scene** specify:

Show title = **Yes**

Title*)

Text = **Time-varying flow in a tube**

Show time = **Yes**

Show step number = **Yes**

*) Parameters **Title > ...** will be available after specifying **Show title = Yes**.

Short description of the project, time and step number will appear in the **View** window.

Also specify displaying in the **Title** values of mass flow on the outlet on the current time and cumulative mass flow. These values will be taken from **Stop criteria** that were created before.

For displaying dynamic values of **Stop criterion #0** and **Stop criterion #1** create two elements in the **User values** array. To do so, select in the **Properties** window of the **3D-scene** root folder the **Title > User values** line and click twice there the screen button **(Append item to the array)**. Child groups of parameters **[0]** and **[1]** will appear in the **User values** array.

Specify parameters in the **Title > User values > [0]** group:

Line begin = **Mass flow at the outlet**

User stopper = **Stop criterion #0**

Line end = **kg/sec**

Specify parameters in the **Title > User values > [1]** group:

Line begin = **Cumulative mass flow at the outlet**

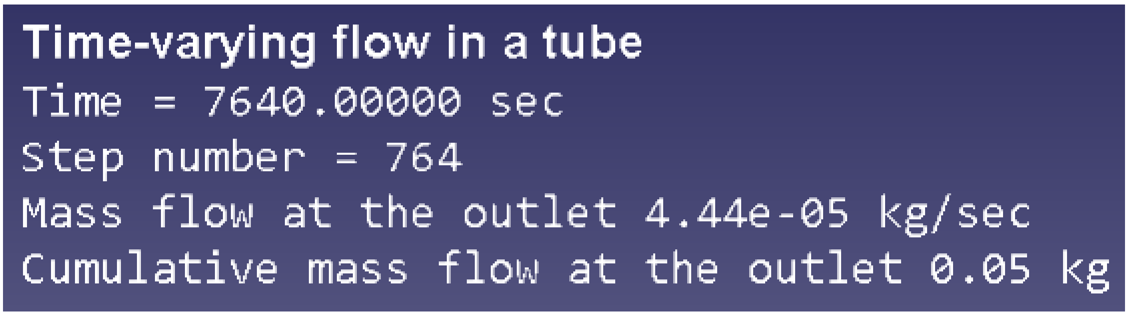
User stopper = **Stop criterion #1**

Line end = **kg**

You can move the **Title** within the **View** window. Change location of the **Title** by specifying in properties of the **3D-scene** root folder:

Title > Location > Horiz. shift = 0.25

Here is an example of the **Title** when the project is running:



4.1.2.7.3 Velocity distribution

- In the **Properties** window of **Plane #0** specify:

Object

Normal

X = **0**

Y = **0**

Z = **1**

(to direct a **Plane**'s normal along the axis **Z**, you can also click the **Operations >** button in the **Properties** window of the **Plane**)

- Create a layer **Vectors** on **Plane #0**
- In the **Properties** window of the **Vectors** specify:

Grid

Size 1 = **50**

Size 2 = **11**

Coloring

Variable

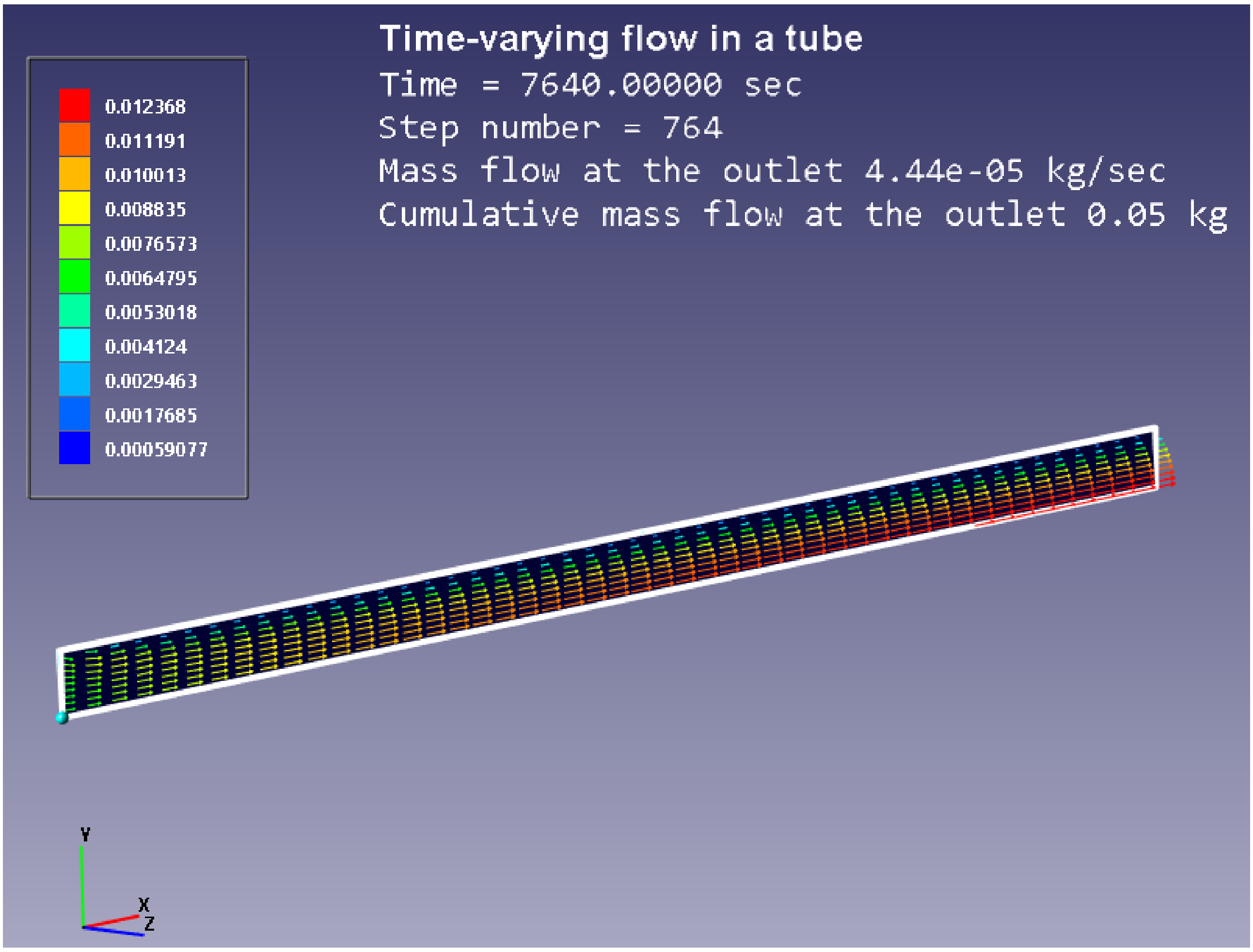
Variable = **Velocity**

Palette

Appearance

Enabled = Yes

The program will automatically specify the variable, which is used to build the vectors, **Variable > Variable = Velocity**.

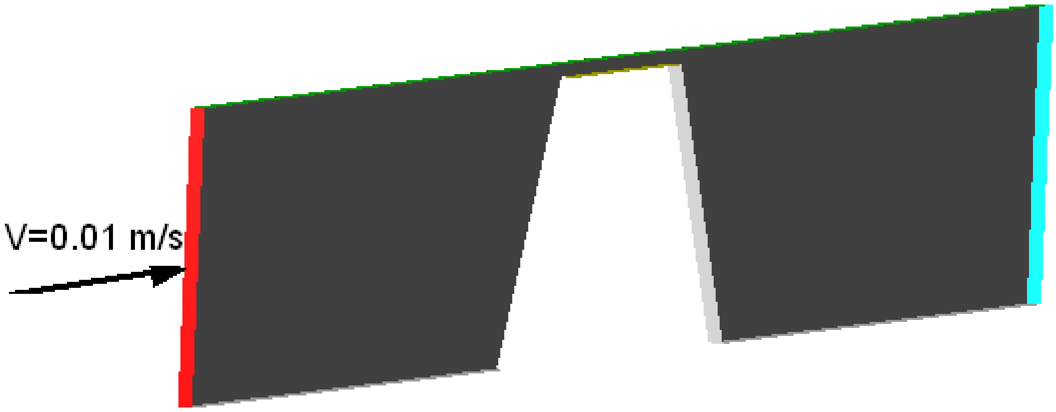


During the computation, the **View** window will display distribution of velocities at the current time moment. As the flow is non-stationary, so the displayed distribution does not correspond to the flow at other time moments. The **View** window will also contain a legend with matching the coloring to values of velocity and a title, which has been specified before, with information about the current mass flow.

4.1.3 Flow in clearance - use of the Gap model

In this example we consider simulating of flow in a narrow two-dimensional channel using the **Gap model**.

The **Gap model** is used in conjunction with a given model of the flow and is designed for taking into account the resistance created by a narrow channel. The **Gap model** avoids resolution of the narrow channel by the grid. The **Gap model** is only applied in cells of the gap. The gap cells are cells, which locate between two 'gap-bounding' surfaces. Surfaces are 'gap-bounding' when the distance between them does not exceed the specified maximal gap clearance. The gap cells are identified by *FlowVision* automatically.



Geometry	`Gap_Channel.wrl`
Project	`Gap_Channel`

4.1.3.1 Physical model

Do the following steps in the project tree in the **Preprocessor** tab.

In the folder **Substances**:

- Create **Substance #0**.
- Specify the following properties of **Substance #0**:

Aggregative state		**= Liquid**	
Molar mass			
	Value	**= 0.018**	[kg mole^{-1}]
Density			
	Value	**= 1000**	[kg m^{-3}]
Viscosity			
	Value	**= 0.001**	[kg m^{-1}s^{-1}]
Specific heat			
	Value	**= 4217**	[J kg^{-1} K^{-1}]

In the folder **Phases**:

- Create a continuous **Phase #0**.
- In **Phase #0** add **Substance #0** into the folder **Substances**.
- Specify properties of **Phase #0 > Physical processes**:

Motion = **Navier-Stokes model**

In the folder **Models**:

- Create **Model #0**.
- Add **Phase #0** into subfolder **Model #0 > Phases**
- Specify in the **Properties** window of **Model #0**:

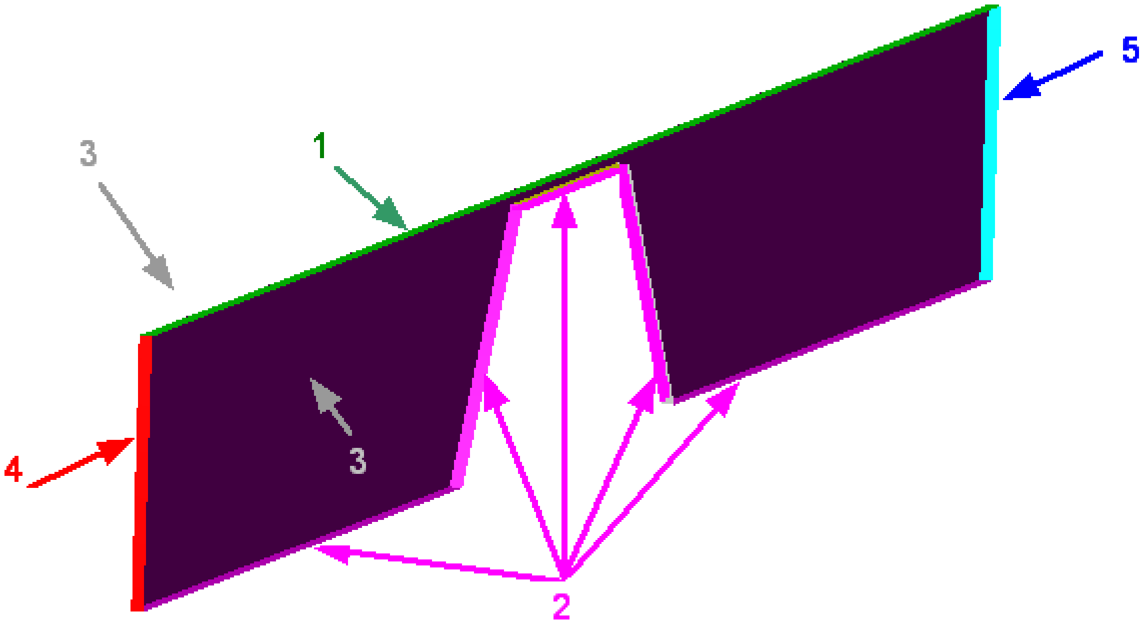
Use Gap model **= Standard Gap model**

4.1.3.2 Boundary conditions

In the **Properties** window of the **SubRegion #0**, specify:

Model = Model #0

Specify the following boundary conditions:



Boundaries 1 and 2
(You have to create two different boundary conditions, both of them has type "**Wall**")
Boundary 1 is the upper wall, and boundary 2 is the lower wall.

Type		**= Wall**
Variables		
	Velocity (Phase #0)	**= No slip**

Boundary 3 (set on the large side faces)

Type		**= Symmetry**
Variables		
	Velocity (Phase #0)	**= Slip**

Boundary 4 (input)

Type		**= Inlet/Outlet**
Variables		
	Velocity (Phase #0)	**= Normal mass velocity**

Specify the numerical value of the **Mass velocity**:

Velocity (Phase #0)

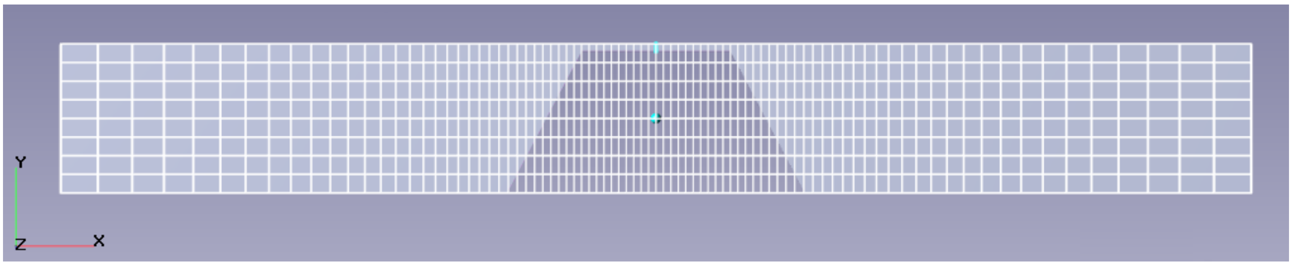
Mass velocity	= **10**	[kg m^{-2} s^{-1}]

Boundary 5 (outlet)

Type		**= Free Outlet**	
Variables			
	Velocity (Phase #0)	**= Pressure**	
	Value	**= 0**	[Pa]

4.1.3.3 Initial grid

In this example in order to better resolve the flow in the channel area, it is necessary to specify a two-dimensional non-uniform computational grid, condensed near the channel.



In the **Properties** window of the **Initial grid**, click the button to open the **Initial grid editor**.

Specify in the **Initial grid editor**:

for axis OX

Grid parameters

h_max = **0.00025** [m]

h_min = **0.00005** [m]

Insert reference lines with coordinates:

x1 = 0.0035 [m]

x2 = 0.0045 [m]

Specify **Reference line parameters** for the reference line with coordinate **x=0**:

h = **0.00025** [m]

Specify **Reference line parameters** for the reference line with coordinate **x=0.0035**:

h = **0.00005** [m]

Specify **Reference line parameters** for the reference line with coordinate **x=0.0045**:

h = **0.00005** [m]

Specify **Reference line parameters** for the reference line with coordinate **x=0.008**:

h = **0.00025** [m]

Click **OK** to close the **Initial grid editor** with saving the entered data.

Specify in the **Properties** window of the **Initial grid**:

Grid structure = **2D**

Plane = **XY**

nY = **8**

If we used adaptation of the computational grid in this exercise, then the settings **Grid structure = 2D** and **Plane = XY** would prevent the adaptation along the axis Z (these settings mean that the grid will always contain only one cell along Z). As no adaptation is used in this exercise, you can just set **nZ=1**.

In the **Properties** window of the **Initial grid** click **Apply**.

4.1.3.4 Parameters of calculation

Specify in the **Solver** tab in properties of the **Time step** element:

Method = **In seconds**

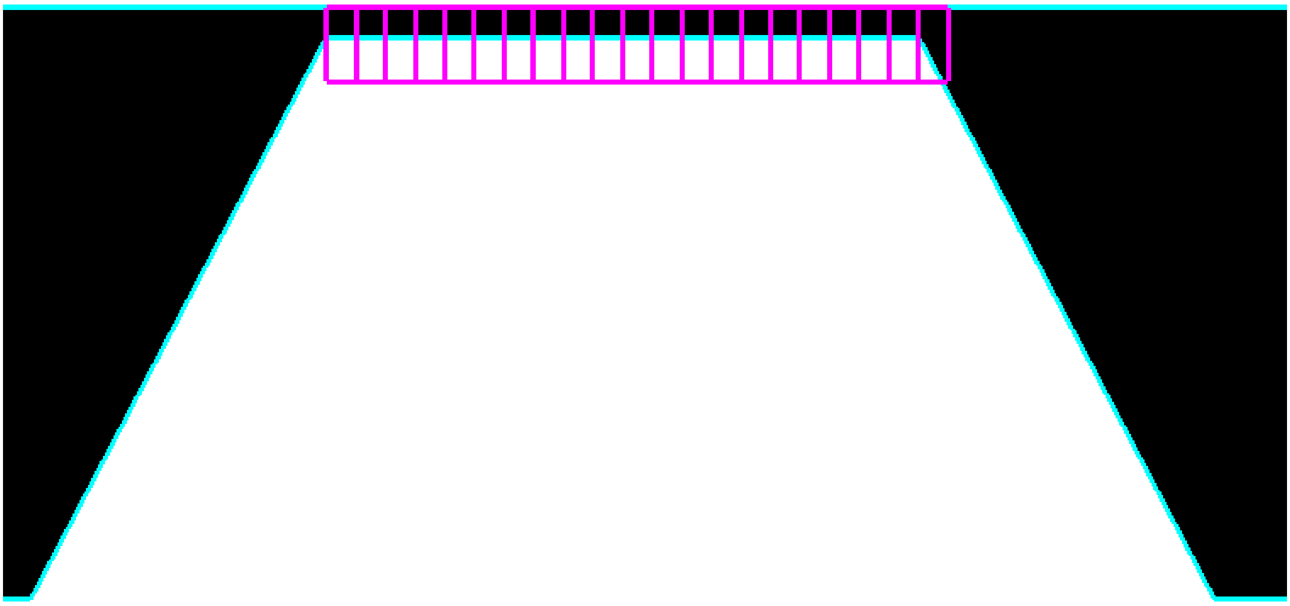
Constant step = **0.001** [s]

4.1.3.5 Visualization

To view the dynamics of the solution during the computation, prior to computation, specify visualization of:

1. Distribution of gap cells
2. Velocity distribution in the plane of the flow

4.1.3.5.1 Distribution of gap cells



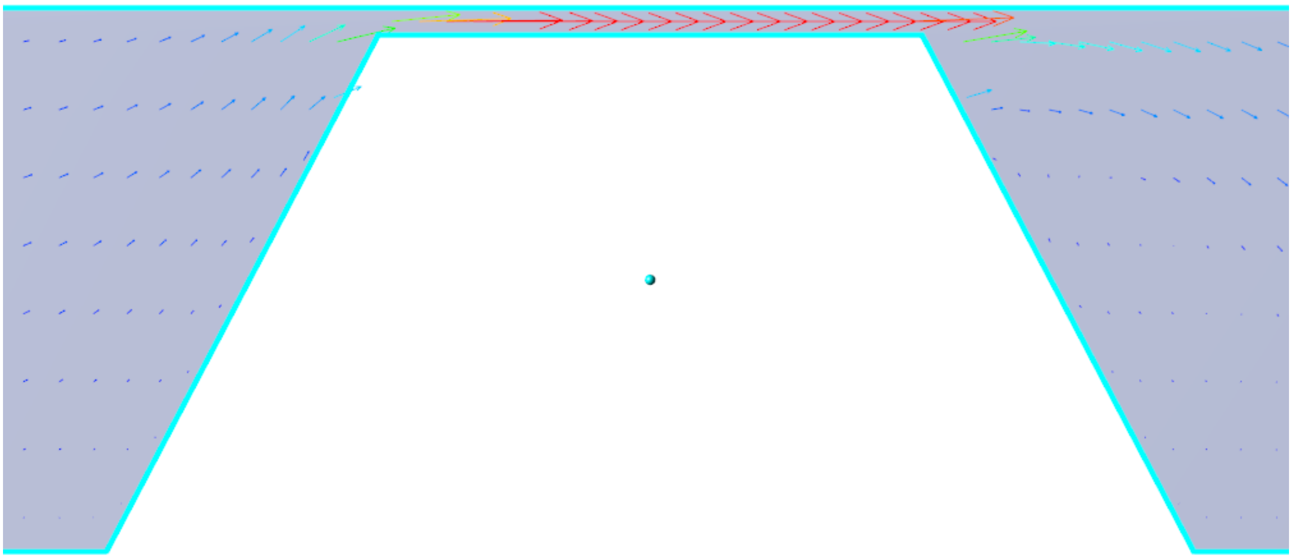
- Create a layer **Cell set** on the **Computational space**
- In the **Properties** window of this layer **Cell set** specify:

Type = **Gap**

Appearance

Mode = **Lines**

4.1.3.5.2 Velocity distribution



- In the **Properties** window of **Plane #0** specify:

Object

  Normal

    X = **0**

    Y = **0**

    Z = **1**

(to direct a **Plane**'s normal along the axis **Z**, you can also click the **Operations >** button in the **Properties** window of the **Plane**)

- Create a layer **Vectors** on **Plane #0**
- In the**Properties** window of the layer **Vectors** specify:

On regular grid = **No**

Coloring

  Variable

    Variable = **Velocity**

  Value range

    Mode = **Manual**

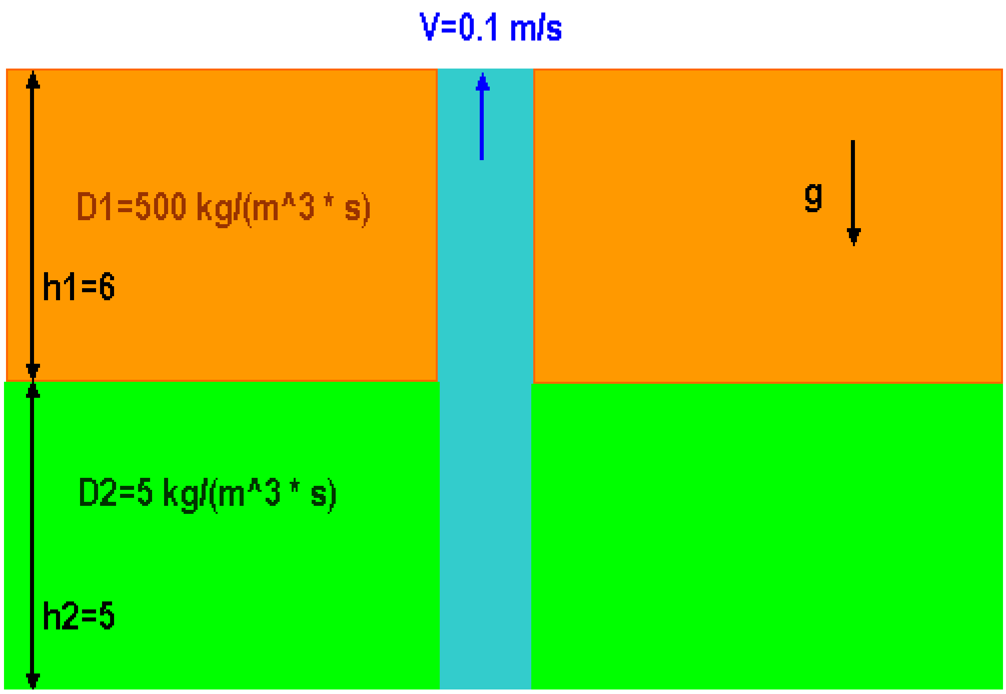
    Max = **0.2**

    Min = **0**

The program will automatically specify the variable, which is used to build the vectors, **Variable > Variable = Velocity**.

4.1.4 Flow of crude oil in a petroleum reservoir

A model of the movement of oil in the reservoir with influence of gravity. From a wellbore with radius **r** oil is pumped at a speed **V**. The upper layer with thickness **h1** has larger resistance **D1** and almost does not allow the oil to go through. The lower layer has thickness **h2** and lower resistance **D2** and has a better capacity.



Dimensions:

Radius of the wellbore	r	= 1	[m]

Thickness of the upper layer	h1	= 6	[m]
Thickness of the lower layer	h2	= 5	[m]
Velocity of the flow in the wellbore:			
Velocity:	V	= 0.1	[m s^{-1}]
Parameters of the substance:			
Density	ρ	= 1000	[kg m^{-3}]
Viscosity	μ	= 0.01	[kg $m^{-1}s^{-1}$]
Geometry:	`Oil.STL`		
Project:	`Oil`		

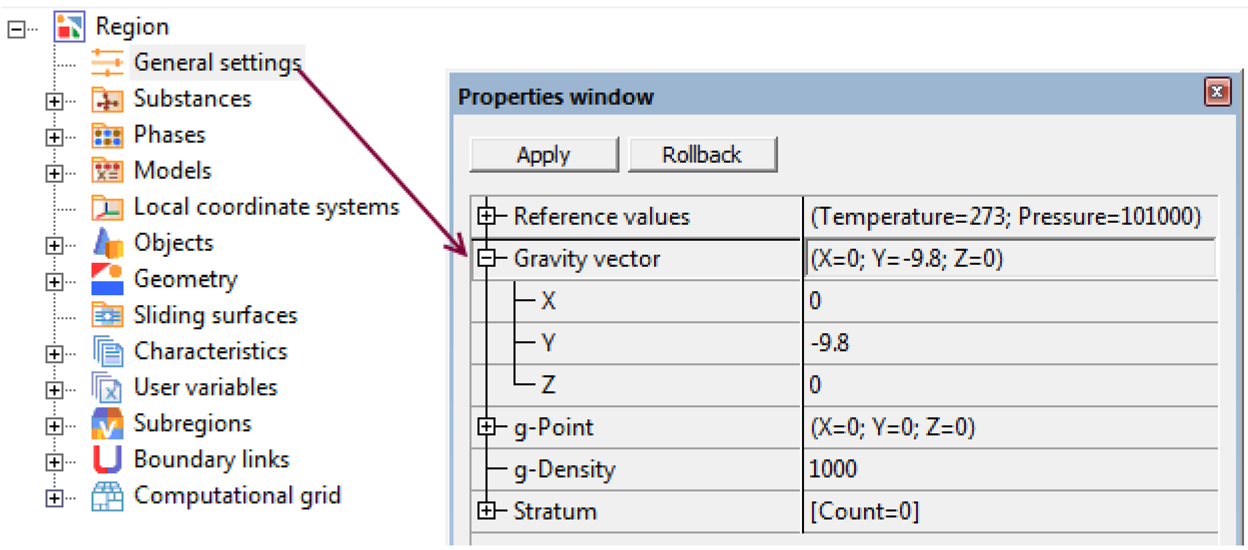
4.1.4.1 Physical model

In this problem, motion of the fluid in a gravitational field is simulated. Therefore, you have to specify the gravity vector. Also, problems in which the motion of the fluid in a gravitational field is simulated, it is possible to count the pressure in the computational domain not from the reference pressure, but from the equilibrium hydrostatic pressure. This allows you to improve the accuracy of the computation and specify in the initial and boundary conditions not the pressure with hydrostatic column, but the difference between the real pressure and the equilibrium pressure. For this the hydrostatic parameters are to be specified.

Do the following steps in the project tree in the **Preprocessor** tab.

In properties of the **General settings** specify:

Gravity vector			
X	**= 0**		[m s^{-2}]
Y	**= -9.8**		[m s^{-2}]
Z	**= 0**		[m s^{-2}]
g-Point			
X	**= 0**		[m]
Y	**= 0**		[m]
Z	**= 0**		[m]
g-Density	**= 1000**		[kg m^{-3}]



In the folder **Substances**:

- Create **Substance #0**
- Specify the following properties of the **Substance #0**:

Aggregative state	**= Liquid**	
Molar mass		
Value	**=0.018**	[kg $mole^{-1}$]
Density		
Value	**= 1000**	[kg m^{-3}]
Viscosity		
Value	**= 0.01**	[kg $m^{-1}s^{-1}$]
Specific heat		
Value	**= 4217**	[J kg^{-1} K^{-1}]

To simulate the oil, we use in this example a **Substance** with properties of water except much larger viscosity.

In the folder **Phases**:

- Create a continuous **Phase #0.**
- In **Phase #0** add **Substance #0** into the folder **Substances**.
- Specify properties of **Phase #0 > Physical processes**:

Motion **= Navier-Stokes model**

In the folder **Models**:

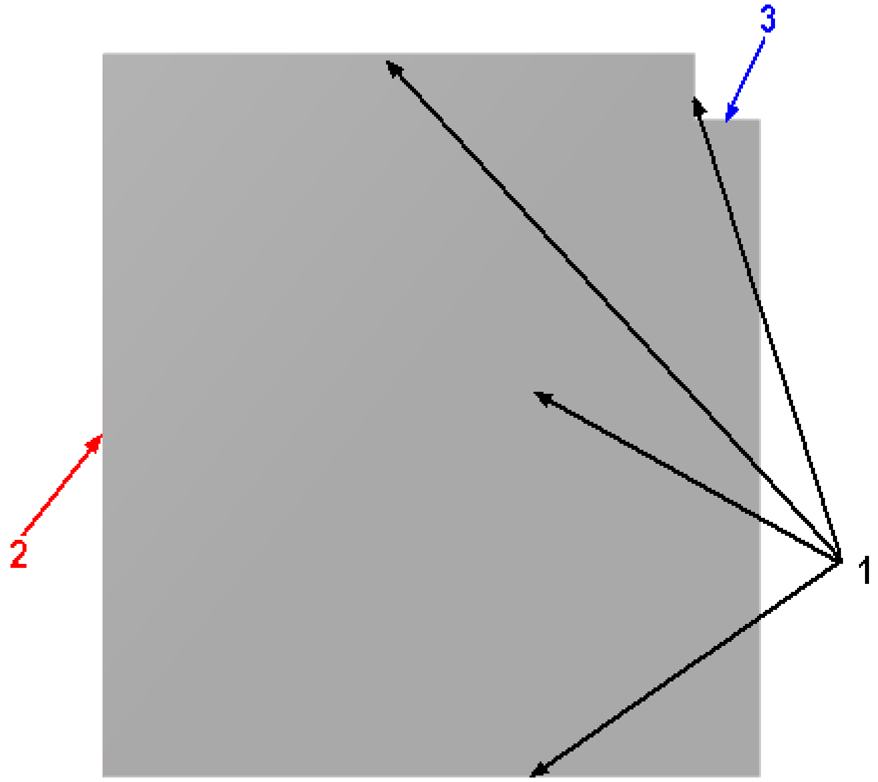
- Create **Model #0**
- Add **Phase #0** into subfolder **Model #0 > Phases**

4.1.4.2 Boundary conditions

In the **Properties** window of the **SubRegion #0**, specify:

Model = Model #0

Specify the following boundary conditions:



Boundary 1

Name		**= Symmetry**
Type		**= Symmetry** *)
Variables		
	Velocity (Phase #0)	**= Slip**

Boundary 2

Name		**= Inlet**	
Type		**= Inlet/Outlet**	
Variables			
	Velocity (Phase #0)	**= Total pressure**	
	Value	**= 0**	[Pa]

Boundary 3

Name		**= Outlet**
Type		**= Inlet/Outlet**
Variables		
	Velocity (Phase #0)	**= Normal mass velocity**

Specify the numerical value of the **Mass velocity**:

Velocity (Phase #0)

Mass velocity	= **-100**	[kg m^{-2} s^{-1}]

Note:

*) In order to set the boundary condition on the boundary corresponding to the slip without leakage, use the **Symmetry** type of the boundary condition.

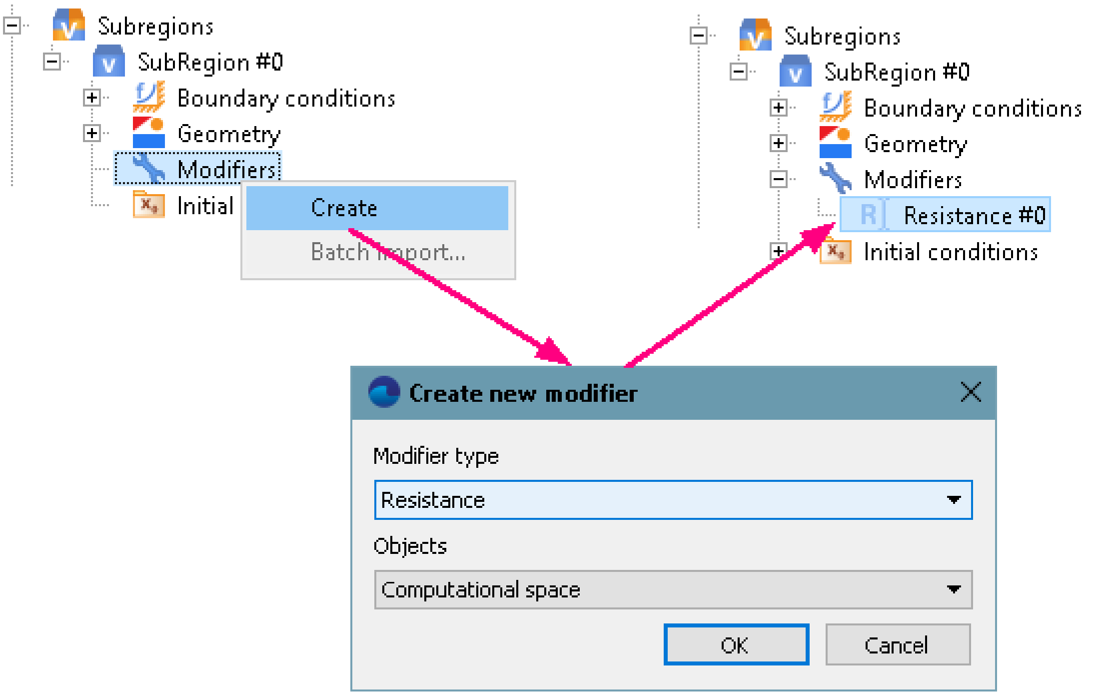
4.1.4.3 Modifiers

Modifiers are **Subregion**'s elements that allow modifying the geometry and/or the solution in the area of their application. In particular, you can use modifiers to specify a volume force, resistance, heat sources, etc.

The process of specifying a **Modifier** consists of two stages:

- specifying the area where the **Modifier** is applied
- specifying the **Modifier** on/in this area

In this example, we need two resistances and, accordingly, we will specify two **Modifiers**.



In order to set the modifier, which corresponds to the main resistance of the wellbore do the following steps:

- In **SubRegion #0** in the context menu of folder **Modifiers**, select **Create**.
- In the **Create new modifier** window, which opens, select:

Modifier type = **Resistance**

Objects = **Computational space**

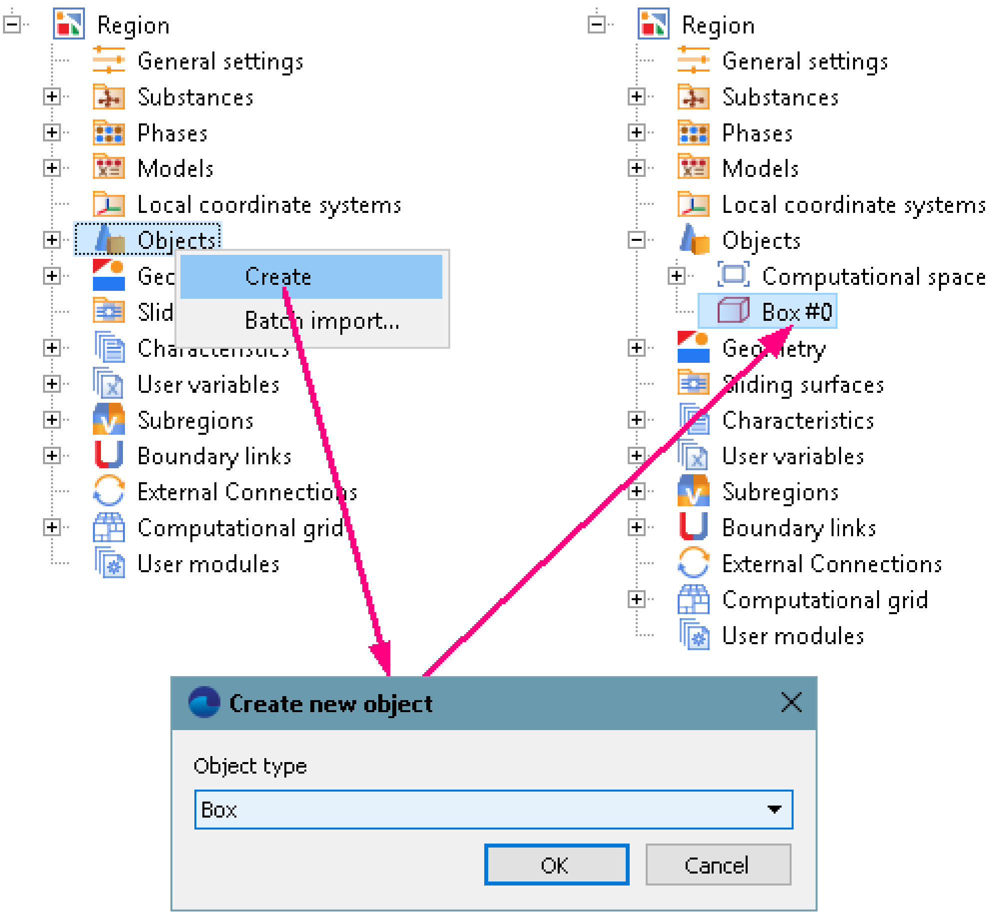
- In properties of the new modifier **Resistance #0** specify:

Activation

Type = **Permanent**

Resistance coef. = **5**

In order to set the modifier, which corresponds to the resistance of the reservoir in the middle of the computational domain, it is necessary to create not only the modifier properties, but also an **Object** on which the modifier will be applied.



To specify the **Object** do the following:

- From the context menu of the folder **Objects**, select **Create**.
- In the **Create new object** window that appears, select **Box**.
- In the **Properties** window of the **Box #0** specify:

Location

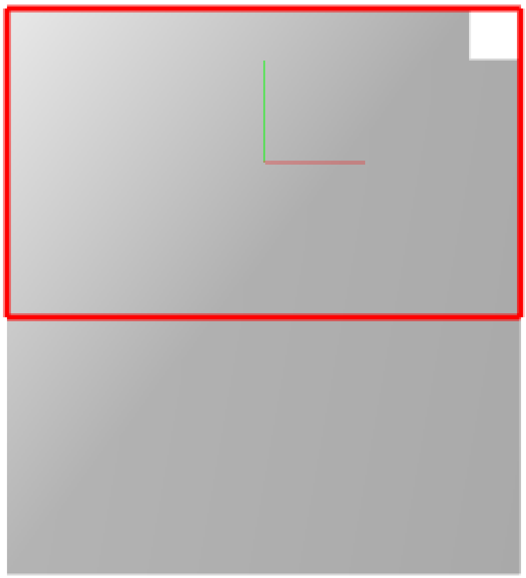
Reference point

X	**= -5**	[m]
Y	**= -3**	[m]
Z	**= 0**	[m]

Size

X	**= 10**	[m]
Y	**= 6**	[m]
Z	**= 0.35**	[m]

In the **View** window an image of **Box #0** will appear.



To create a **Resistance** inside of the created **Object** do the following:

- In **SubRegion #0**, from the context menu of the folder **Modifiers**, select **Create**.
- In the **Create new modifier** window, which opens, specify:

Modifier type	**= Resistance**
Objects	**= Box #0**

- In properties of the new modifier **Resistance #1**, which appears in the project tree, specify:

Activation	
Type	**= Permanent**
Resistance coef.	**= 500**

To specify absence of the resistance in the borewell itself, create a modifier with zero resistance on an **Object**, which corresponds to the borewell.

To specify the **Object** do the following:

- From the context menu of the folder **Objects**, select **Create**.
- In the **Create new object** window that appears, select **Box**.
- In the **Properties** window of **Box #1** specify:

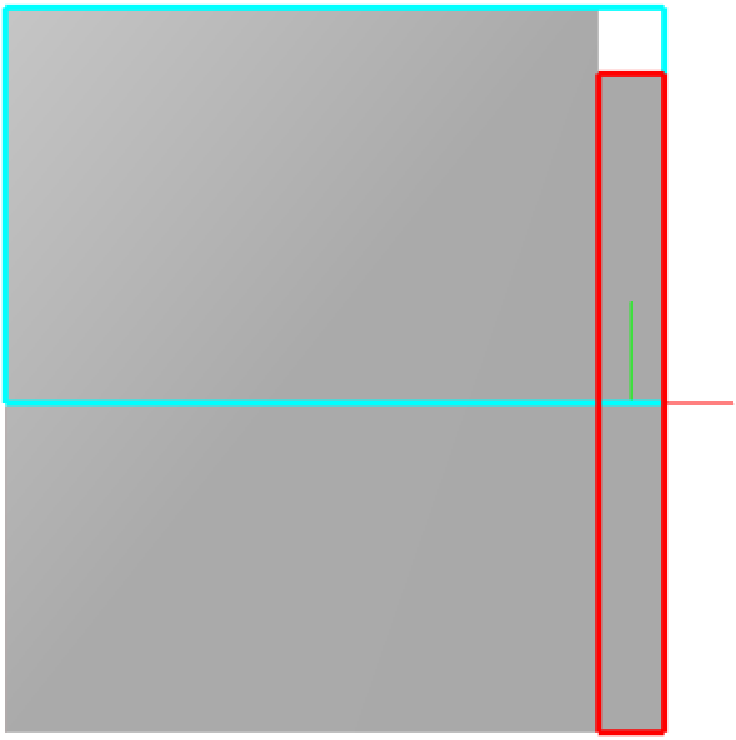
Location

Reference point			
	X	**= -0.5**	[m]
	Y	**= -6**	[m]
	Z	**= 0**	[m]

Size

X	**= 1**	[m]
Y	**= 10**	[m]
Z	**= 0.35**	[m]

In the **View** window an image of **Box #1** will appear.



To create a **Resistance** inside the created **Object** do:

- In **SubRegion #0** in the context menu of the **Modifiers** folder, select **Create**.
- In the **Create new modifier** window, which appears, specify:

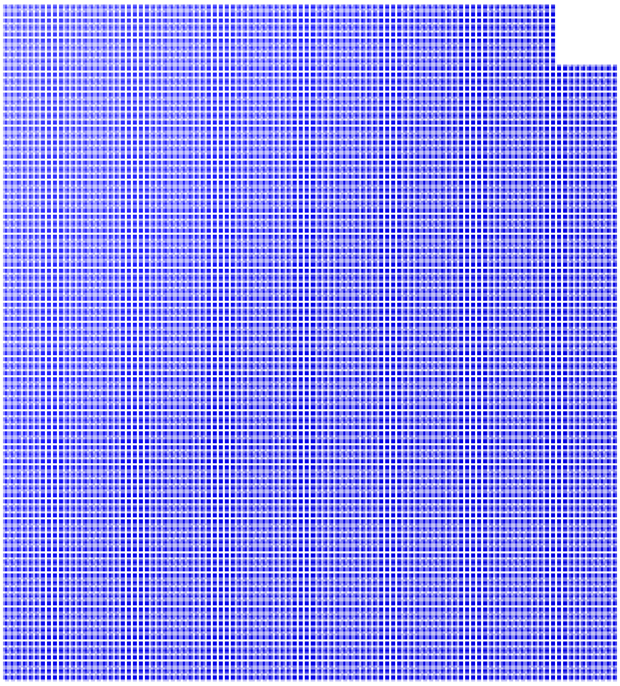
Modifier type	**= Resistance**
Objects	**= Box #1**

- In properties of the new modifier **Resistance #2**, which appears in the project tree, specify:

Activation	
Type	**= Permanent**
Resistance coef.	**= 0**

Modifiers of the same type are applied sequentially in order of their location in the folder **Modifiers**. Priority of a **Modifier** is higher when it is located lower in the list. Thus, in the area, which is the set-theoretic intersection of **Box #0**, **Box #1**, and **SubRegion #0**, the **Modifier** is applied, which is specified in **Box #1**.

4.1.4.4 Initial grid



Specify in the **Properties** window of the **Initial grid**:

Grid structure = 2D

Plane = XY

nX = 100

nY = 100

In the **Properties** window of the **Initial grid** click **Apply**.

4.1.4.5 Parameters of calculation

Specify in the **Solver** tab in properties of the **Time step** element:

Method	**= Via CFL number**
Convective CFL	**= 1**
Diffusive CFL	**= 1**

4.1.4.6 Stopping conditions

In this exercise, we recommend that you define the stopping condition by the static pressure at the outlet.

Create **Characteristics**:

- In the **Preprocessor** tab, create a **Supergroup** on the surface of selecting the boundary condition **Outlet** using the command **Create supergroup > In Preprocessor** from the context menu of the boundary condition.
- Create **Characteristics** for the just created **Supergroup on "Outlet"**.
- In the **Properties** window of **Characteristics #0** specify:

Variable

Variable	**= Pressure**

Specify the stopping criterion:

- in the **Solver** tab, create **Stop criterion #0** in the folder **Stopping conditions > User values** (select **Create** from the folder's context menu).

- In the **Properties** window of the new just created **Stop criterion #0** set:

Level	**= 1e-6**
Averaging	**= By period**
Period	
Type	**= By time**
Number of seconds	**= 100**
Object	**= Characteristics #0 (Supergroup on "Outlet")**
Variable	**= <f surf>**

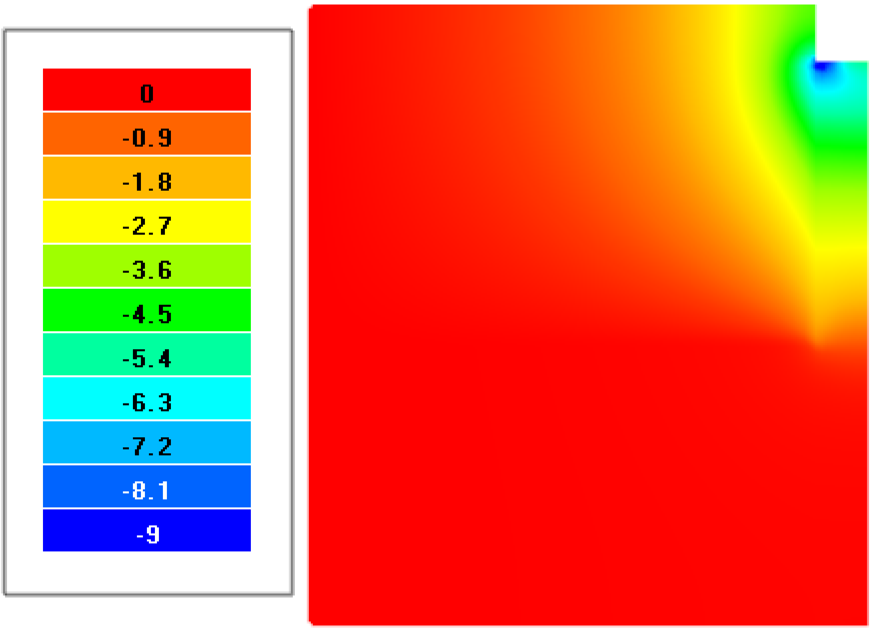
4.1.4.7 Visualization

Specify visualization of:

1. [Pressure distribution]() in the plane of the flow
2. [Pressure distribution with hydrostatic taken into account]() in the plane of the flow

4.1.4.7.1 Pressure distribution

Visualization of pressure distribution when the step number is near 500:



- In the **Properties** window of **Plane #0** specify:

Object	
Normal	
X	**= 0**
Y	**= 0**
Z	**= 1**

(to direct a **Plane**'s normal along the axis **Z**, you can also click the **Operations >** button in the **Properties** window of the **Plane**)

- Create a layer **Color contours** on **Plane #0**.
- In the **Properties** window of **Color contours #0** specify:

Variable	
Variable	**= Pressure**
Value range	
Mode	**= Manual**

Max = 0

Min = -9

Palette

Appearance

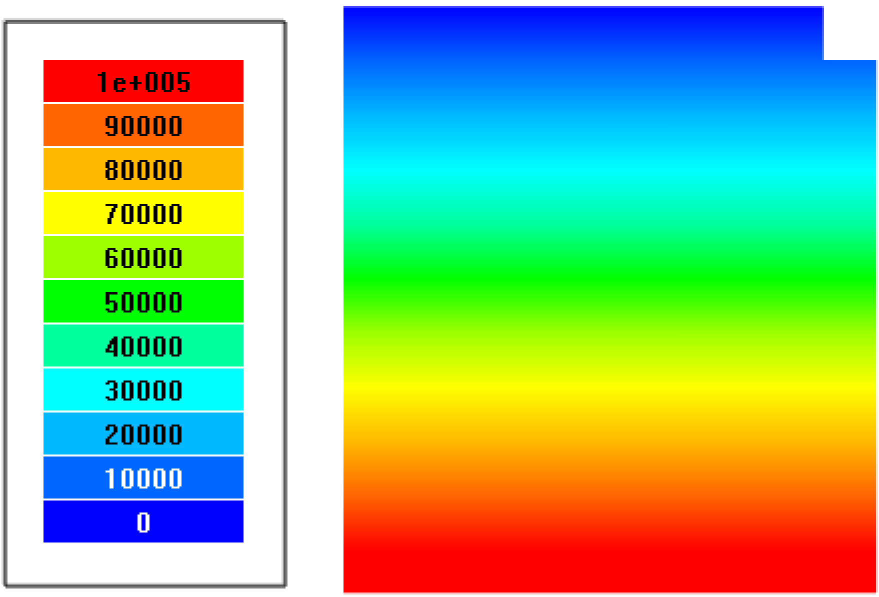
Enabled = Yes

Style = Style 1

Note:

When hydrostatic is set, the **Pressure** variable corresponds to the deviation from the equilibrium hydrostatic pressure. To display pressure distribution relative to the reference pressure, without deduction of the equilibrium hydrostatic pressure, you have to display pressure distribution with hydrostatic taken into account.

4.1.4.7.2 Pressure distribution with hydrostatics taken into account



- Create a layer **Color contours** on **Plane #0**.
- In the **Properties** window of **Color contours #1** specify:

Variable

Variable = Pressure (+ hydrostatic)

Value range

Mode = Manual

Max = 100000

Min = 0

Palette

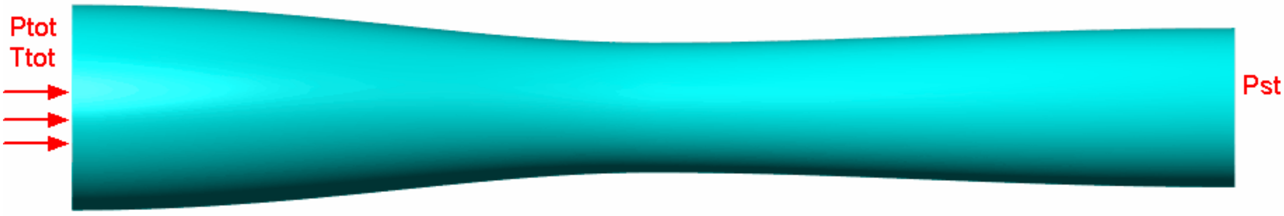
Appearance

Enabled = Yes

Style = Style 1

4.1.5 Transonic flow in Laval nozzle

An one-dimensional transonic flow of inviscid zero-conductivity gas in the Laval nozzle is considered in the given example.



Parameters of the fluid:

Density:	ρ_{inl}	= 1.29	[kg m^{-3}]
Viscosity:	μ	= 0	[kg m^{-1}s^{-1}]
Thermal conductivity	λ	= 0	[W m^{-1}K^{-1}]
Specific heat	c_p	= 1009	[J (kg K)$^{-1}$]
Sonic speed	a	= 331.6	[m s^{-1}]

Inlet parameters:

Total pressure on the inlet	P	=6895	[Pa]
Total temperature on the inlet	T	=125	[K]

Outlet parameters

Pressure on the outlet	P_{st}	= 5171	[Pa]

Geometry: `Nozzle.STL`

Project: `Nozzle`

This project can require substantial computational resources and time.

4.1.5.1 Physical model

Do the following steps in the project tree in the **Preprocessor** tab.

In the **Properties** window of the element **General settings** specify:

- Specify the following parameters:

Reference values

Temperature	**= 125**	[K]
Pressure	**= 6895**	[Pa]

In the folder **Substances**:

- Create **Substance #0**
- Specify the following properties of **Substance #0** $^{*)}$:

Aggregative state	**= Gas**	
Molar mass		
Value	**= 0.0289**	[kg mole^{-1}]
Density	**= Ideal gas law**	
Specific heat		
Value	**= 1009**	[J kg^{-1} K^{-1}]

In the folder **Phases**:

- Create a continuous **Phase #0**.
- In **Phase #0** add **Substance #0** into the folder **Substances**.
- Specify properties of **Phase #0 > Physical processes** *):

Motion	**= Navier-Stokes model**
Heat transfer	= **Heat transfer via H**

In the folder **Models**:

- Create **Model #0**
- Add **Phase #0** into subfolder **Model #0 > Phases**

Note:

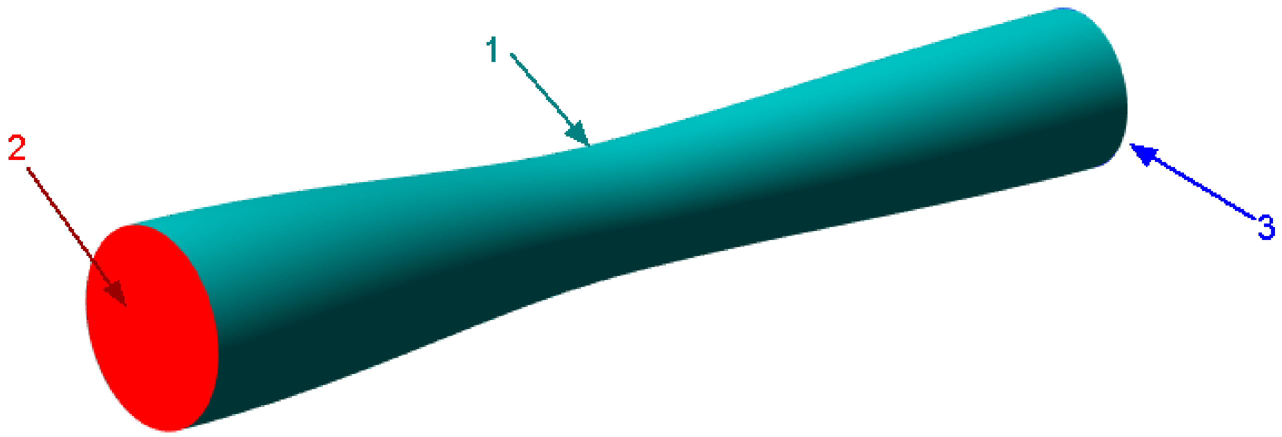
*) When simulating the gas flow at Mach > 0.15 is necessary to consider the dependency of density on pressure. To do this:

1. In properties of the **Substance** you have to specify **Aggregative state = Gas**, the dependency of **Density** by **Ideal gas law** and values of **Molar mass** and **Specific heat**.
2. In **Physical processes** enable computation of the **Heat transfer** equations (select the **Heat transfer via H** option).

4.1.5.2 Boundary conditions

In the **Properties** window of the **SubRegion #0**, specify:

Model = Model #0



Specify the following boundary conditions:

Boundary 1

Name	**= Symmetry**
Type	**= Symmetry** *)
Variables	
Temperature(Phase #0)	**= Symmetry**
Velocity(Phase #0)	**= Slip**

Boundary 2

Name	**= Outlet**
Type	**= Inlet/Outlet**
Variables	

Temperature(Phase #0)	**= Total temperature**	
Value	**= 0**	[K]
Velocity(Phase #0)	**= Total pressure**	
Total pressure	**= 0**	[Pa]

Boundary 3

Name	**= Inlet**	
Type	**= Inlet/Outlet**	
Variables		
Temperature(Phase #0)	**= Temperature**	
Value	**= -10**	[K]
Velocity(Phase #0)	**= Inlet pressure**	
Pressure	**= -1724**	[Pa]

Note:

*) In order to set on a boundary a boundary condition corresponding to the slip without leakage, you have to use the **Symmetry** type of boundary condition.

4.1.5.3 Initial grid

Specify in the **Properties** window of the **Initial grid**:

Grid structure = 1D

Direction = X

nX = 1000

If we used adaptation of the computational grid in this exercise, then adaptation along the non-computational directions **Y** and **Z** would not be done, and the grid would always have only one cell along these directions.

As no adaptation is used here, you can just set **nY=1** and **nZ=1**.

In the **Properties** window of the **Initial grid** click **Apply**.

4.1.5.4 Parameters of calculation

Specify in the **Solver** tab of the **Project** window:

- In the **Properties** window of the element **Time step** specify:

Method	**= Via CFL number**	
Convective CFL	**= 10**	
Max step	**= 0.001**	[s]

- In the **Properties** window of the element **Advanced settings** specify:

Numerical method > Type of scheme	**= Implicit**
Numerical method > Use SGA	**= No**
Numerical method > Pressure gradient	**= With velocity consideration**

- In the **Properties** window of the element **Limiters > Limiters for calculation > Phase Limiters > Phase #0** specify:

Limiter > Density, min.	**= 0.001**	[kg m^{-3}]
Limiter > Temperature abs, min.	**= 10**	[K]
Limiter > Temperature abs, max.	**= 1000**	[K]
Limiter > Velocity, max.	**= 1000**	[m/s]
Limiter > Pressure abs, min.	**= 0**	[Pa]
Limiter > Pressure abs, max.	**= 10000**	[Pa]

Note:

For simulations with high gradients of velocity, temperature and pressure it is recommended to use phase limiters. The limiters help to avoid the divergence of the solution.Also, for one-dimensional simulations and simulations, which use the gap model, it is recommended to specify **Numerical method > Pressure gradient = With velocity consideration.**

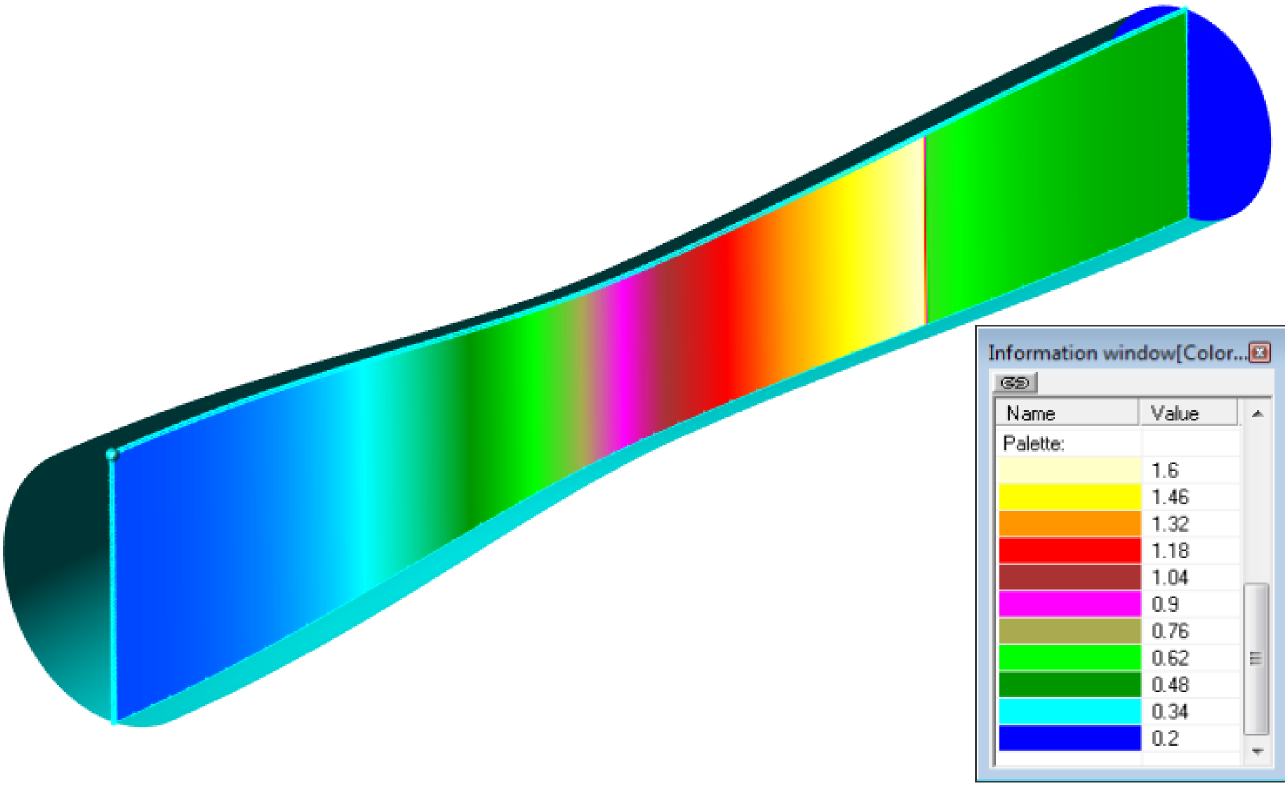
4.1.5.5 Visualization

To view the dynamics of the solution during the computation, prior to computation, specify visualization of:

1. **Mach Number distribution** in the plane of the flow.
2. **Pressure distribution** in the plane of the flow.

4.1.5.5.1 Mach Number distribution

Visualization of the Mach Number distribution when the step number is near 1300:



- In the **Properties** window of **Plane #0** specify:

Object

Normal

X	**= 0**
Y	**= 0**
Z	**= 1**

(to direct a **Plane**'s normal along the axis **Z**, you can also click the **Operations >** button in the **Properties** window of the **Plane**)

- Create a layer **Color contours** on the **Plane**.
- In the **Properties** window specify:

Variable

Variable	**MachNumber**

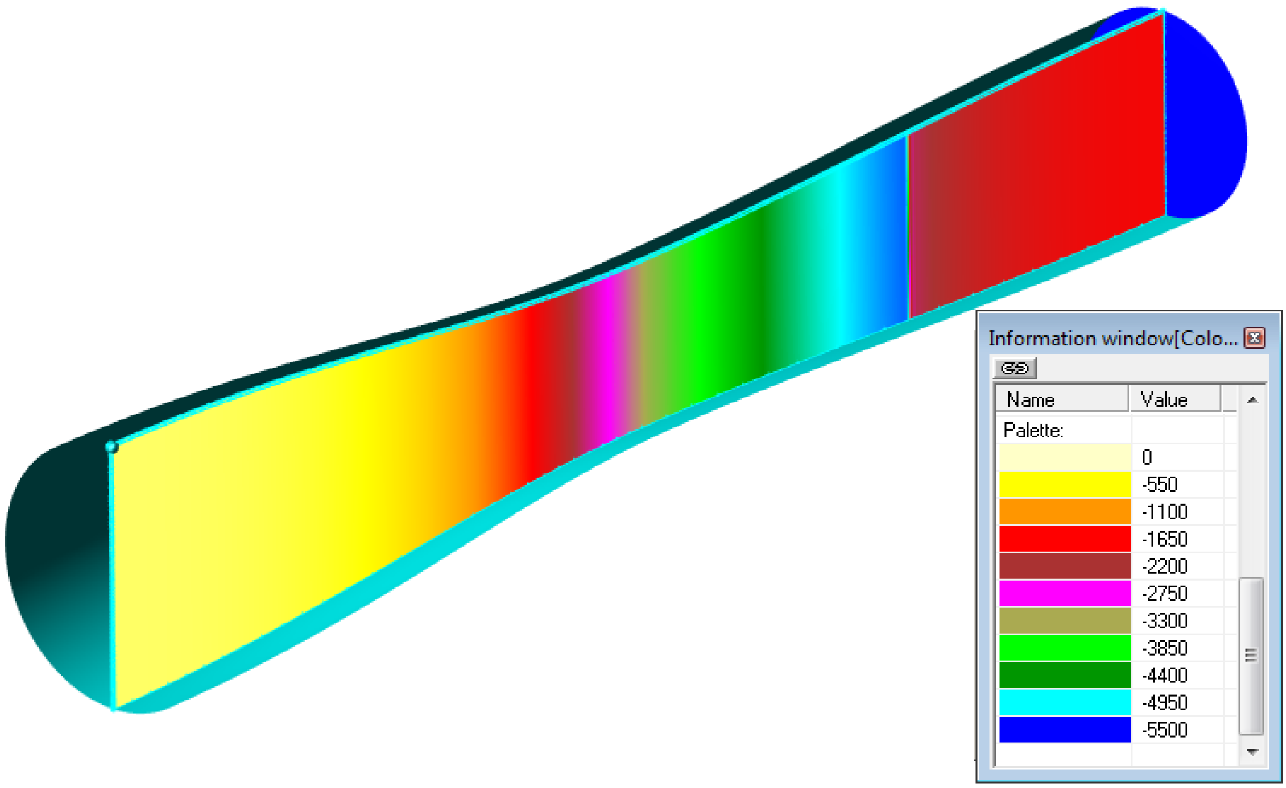
Value range

Mode	**= Manual**
Max	**= 1.6**

Min **= 0.2**

- Load the thermal palette from the file **`heat.fvpal`**:
  - In the **Properties** window of the **Layer** click button **Palette > Operations >** **(Load palette from file)**
  - In the operation system's window, which appears, select the file **`heat.fvpal`** (this file locates in the directory where *FlowVision* is installed).

4.1.5.5.2 Pressure distribution



- Create a layer **Color contours** on the plane.
- In the **Properties** window specify:

Variable

Variable **Pressure**

Value range

Mode **= Manual**

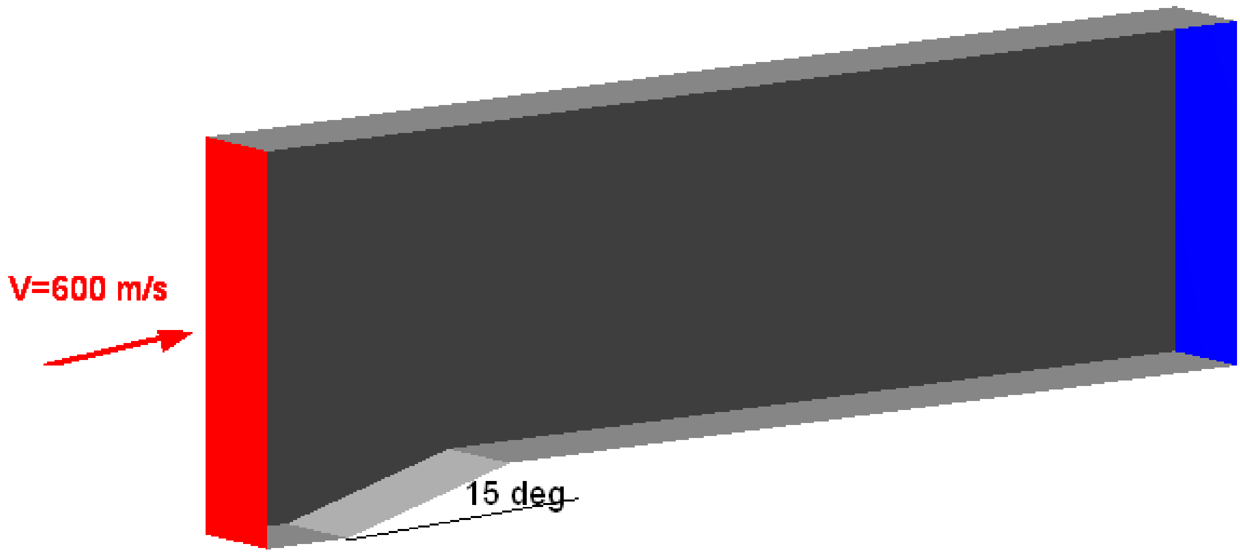
Max **= 0**

Min **= -5500**

- Load thermal palette from the file **`heat.fvpal`** (this file locates in the directory where *FlowVision* is installed).

4.1.6 Supersonic flow past wedge

In this example a supersonic flow around a 15-degree wedge in two-dimensional channel is simulated.



Dimensions:

Angle:	α	= 15	[degree]
Dimensions of the computational domain		6 × 2 × 1	[m × m × m]

Parameters of the substance:

Molar mass:	M	= 0.0289	[kg mole^{-1}]
Viscosity:	μ	= 1.82 10^{-5}	[kg m^{-1}s^{-1}]
Thermal conductivity	λ	= 0.026	[W m^{-1}K^{-1}]
Specific heat capacity	c_p	= 1009	[J (kg K)$^{-1}$]
Sonic speed:	a	= 331.6	[m s^{-1}]

Parameters on inlet:

Static pressure	P	= 101000	[Pa]
Temperature at infinity	T	= 273	[K]
Velocity on inlet	V_{inl}	= 600	[m s^{-1}]
Mach number	M	= 1.8	

Geometry: `Wedge.wrl`

Project: `Wedge`

4.1.6.1 Physical model

Do the following steps in the project tree in the **Preprocessor** tab.

In the folder **Substances**:

- Create **Substance #0**.
- In properties of **Substance #0** specify:

Aggregative state	**= Gas**	
Molar mass		
Value	**= 0.0289**	[kg mole^{-1}]

Density	**= Ideal gas law**	
Viscosity		
Value	**= 1.82e-5** *)	[kg $m^{-1}s^{-1}$]
Thermal conductivity		
Value	**= 0.026**	[W $(m\ K)^{-1}$]
Specific heat		
Value	**= 1009**	[J kg^{-1} K^{-1}]

*) **1.82e-5** is notation for 1.82×10^{-5}.

In the folder **Phases**:

- Create a continuous **Phase #0**.
- In **Phase #0** add **Substance #0** into the folder **Substances**.
- Specify properties of **Phase #0 > Physical processes**:

Motion	**= Navier-Stokes model**
Heat transfer	= **Heat transfer via H**

In the folder **Models**:

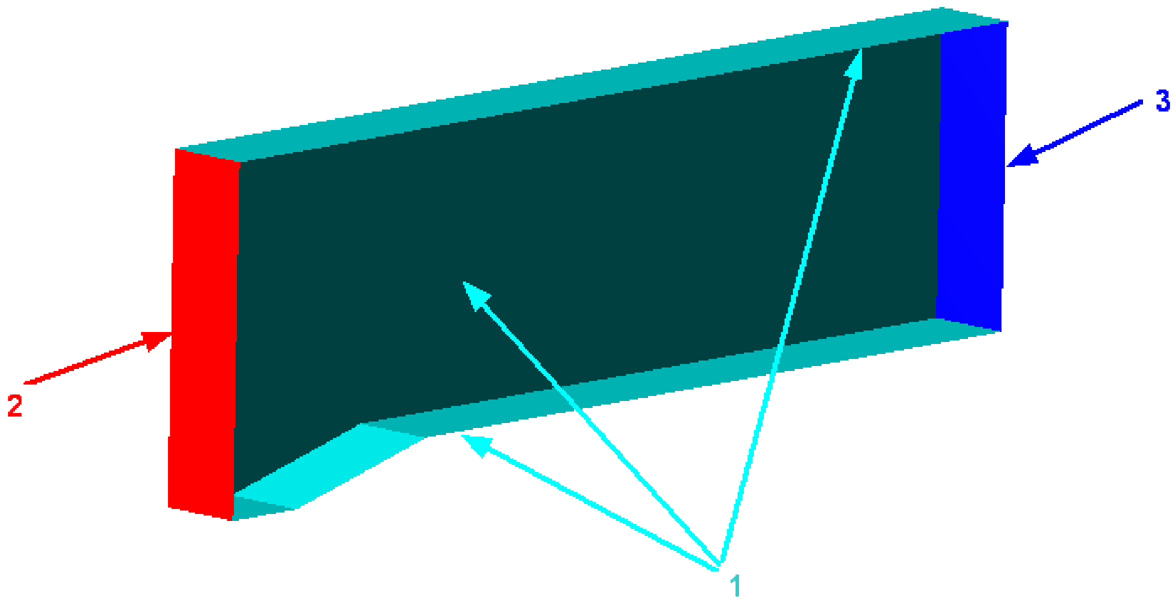
- Create **Model #0**.
- Add **Phase #0** into subfolder **Model #0 > Phases**
- Specify in the folder **Models > Model #0 > Init. data > Init. data #0**:

Velocity (Phase #0)		
X	**= 600**	[m s^{-1}]

4.1.6.2 Boundary conditions

In the **Properties** window of the **SubRegion #0**, specify:

Model = Model #0



Specify the following boundary conditions:

Boundary 1

Type	**= Symmetry**	
Variables		
Temperature(Phase #0)	**= Symmetry**	
Velocity(Phase #0)	**= Slip**	

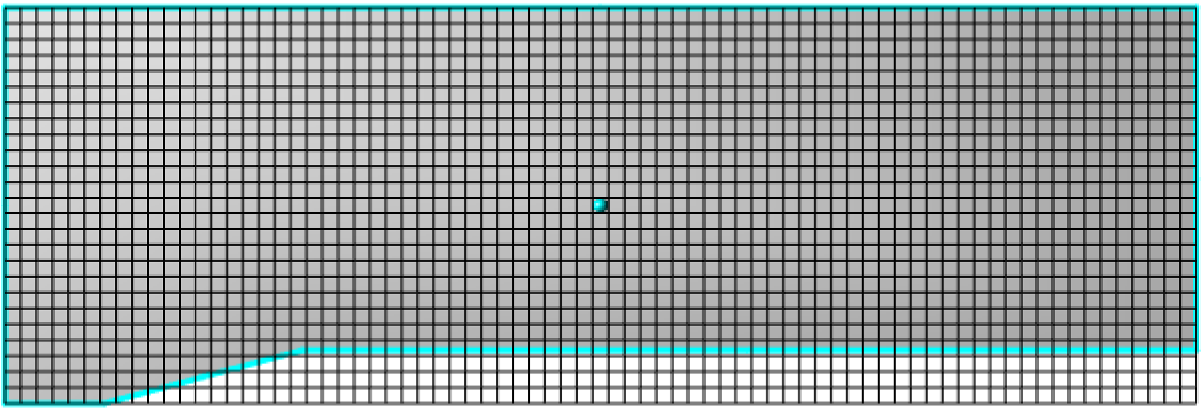
Boundary 2

Type	**= Inlet/Outlet**	
Variables		
Temperature(Phase #0)	**= Temperature**	
Value	**=0**	[K]
Velocity(Phase #0)	**Normal velocity with pressure**	
Velocity	**=600**	[m s^{-1}]
Pressure	**=0**	[Pa]

Boundary 3

Type	**= Free Outlet**	
Variables		
Temperature(Phase #0)	= **Zero gradient**	
Velocity(Phase #0)	**= Pressure**	
Pressure	**=0**	[Pa]

4.1.6.3 Initial grid



Specify properties of the **Initial grid**:

Grid structure = 2D

Plane = XY

nX = 75

nY = 25

In the **Properties** window of the **Initial grid** click **Apply**.

4.1.6.4 Adaptation of the computational grid

In this project it is necessary to solve the grid in the area of pressure surges. To provide this, specify adaptation by the gradient of pressure.

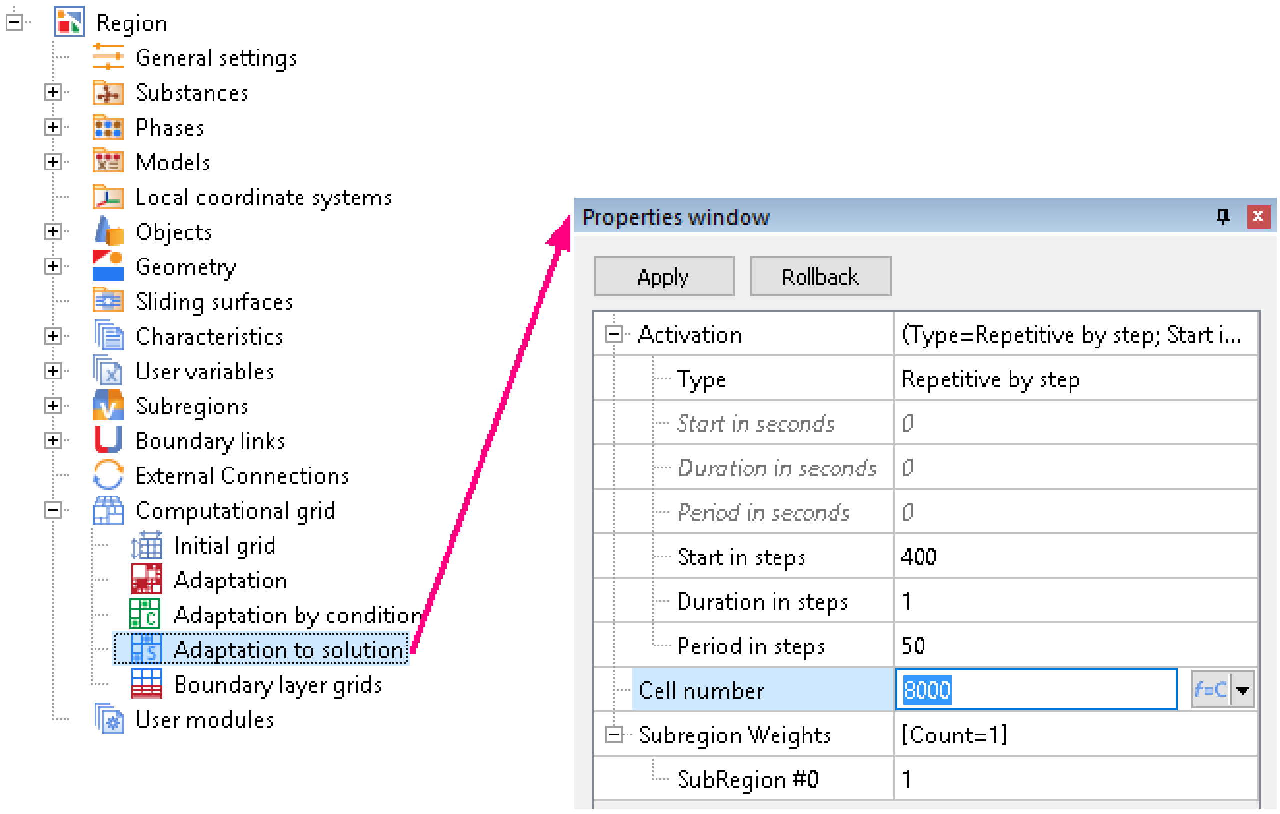
Specify the adaptation by the maximal gradient of pressure:

- In the folder **Computational grid > Adaptation to solution** specify:

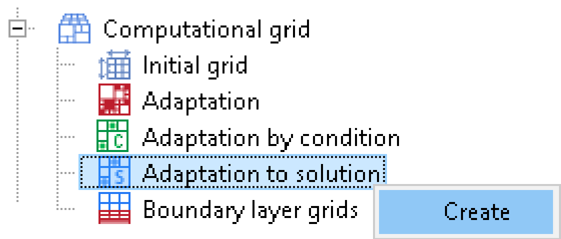
Activation

Type	**= Repetitive by step**
Start in steps	**= 400**
Duration in steps	**= 1**
Period in steps	**= 50**

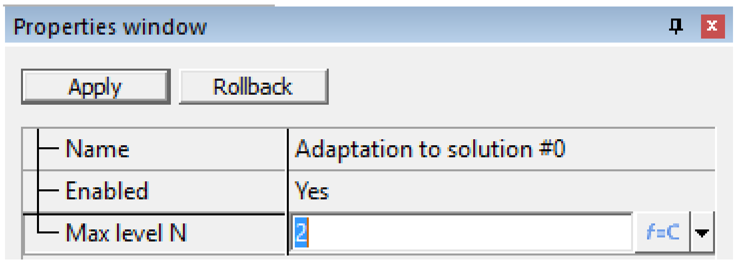
Cell number **= 8000**



- Create in the subfolder **Computational grid > Adaptation to solution** an element **Adaptation to solution #0**.

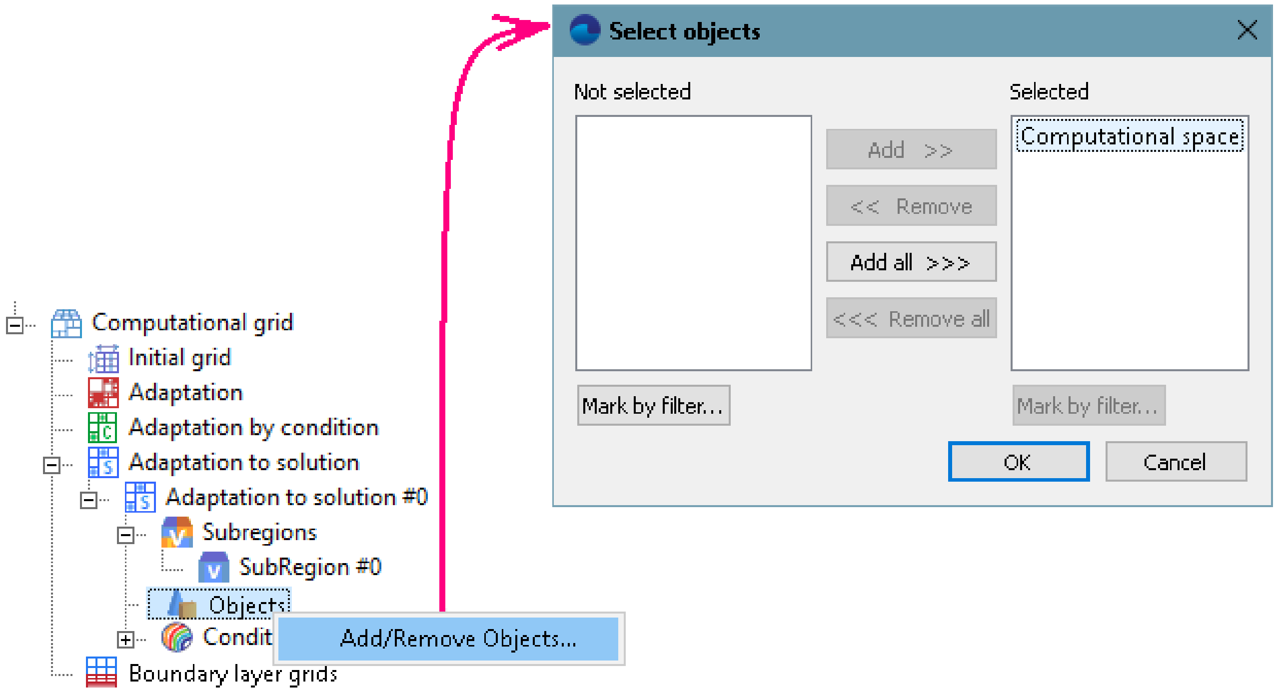


- In the **Properties** window of the just created element **Adaptation to solution #0** specify **Max level N = 2**:



As in this exercise there is only one **Subregion**, you don't have to add **SubRegion #0** into the folder **Computational grid > Adaptation to solution > Adaptation to solution #0 > Subregions** using the **Add/Remove** command from the context menu; **SubRegion #0** is added to this folder automatically at creation of **Adaptation to solution #0**.

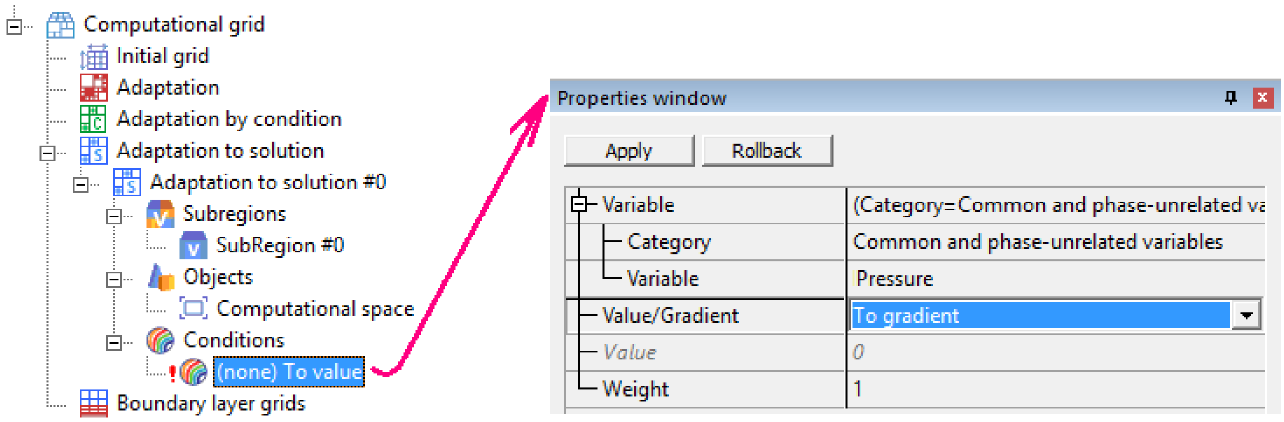
- From the context menu of the subfolder **Computational grid > Adaptation to solution > Adaptation to solution #0 > Objects** select the **Add/Remove Objects** command and in the **Select objects** dialog box, which opens, place **Computational space** to the **Selected** pane and click **OK**:



- In the **Properties** window of the element **Computational grid > Adaptation to solution > Adaptation to solution #0 > (none) To value** specify:

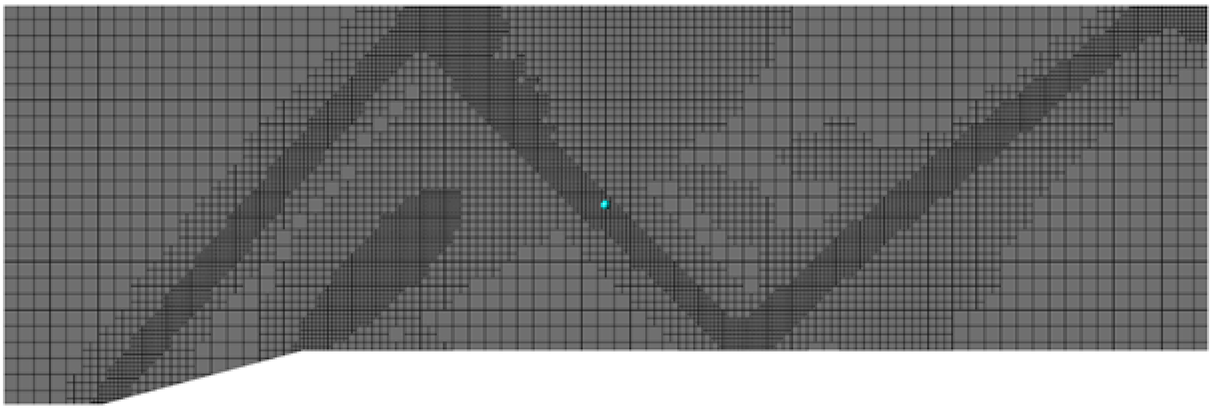
Variable

Variable	**= Pressure**
Value/Gradient	**= To gradient**



After this the child element in the subfolder **Computational grid > Adaptation to solution > Adaptation to solution #0 > Conditions > Conditions** will be presented in the project tree as **Pressure To gradient**.

During the simulation at the 600th time step the following computational grid will be formed:



4.1.6.5 Parameters of calculation

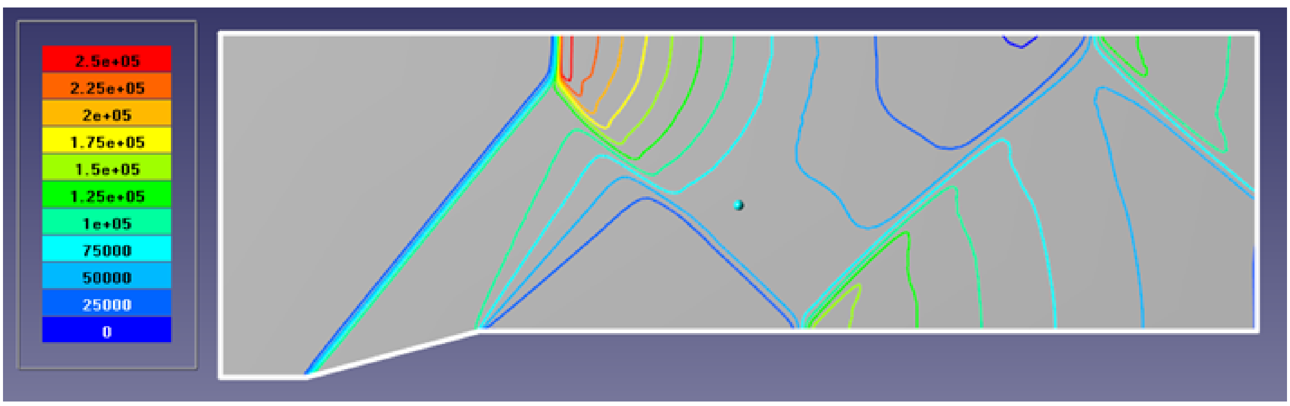
Specify in the **Solver** tab in properties of the **Time step** element:

Method	**= Via CFL number**	
Convective CFL	**= 1**	
Max step	**= 0.0001**	[s]

4.1.6.6 Visualization

To view the dynamics of the solution during the computation, prior to computation, specify visualization of **Pressure distribution** in the plane of the flow before the start of computation.

4.1.6.6.1 Pressure distribution



- In the **Properties** window of **Plane #0** specify:

Object

Normal

X	**= 0**
Y	**= 0**
Z	**= 1**

(to direct a **Plane**'s normal along the axis **Z**, you can also click the **Operations >** Z↓ button in the **Properties** window of the **Plane**)

- Create a layer **Color contours** on the plane.
- In the **Properties** window of the layer specify:

Variable

Variable	**= Pressure**

Value range

Mode	**= Manual**
Max	**= 2.5e5**
Min	**= 0**
Method	**= Isolines**

Palette

Appearance

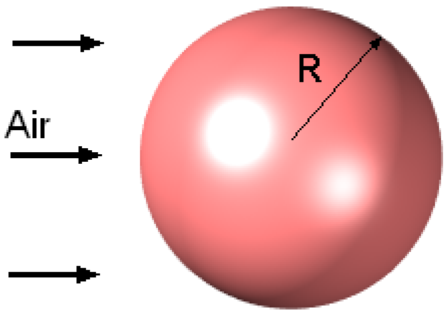
Enabled	**= Yes**
Style	**= Style 1**

Notes:

- In order to view the computational grid with the adaptation, create a **Computational grid section** layer on a **Plane**, which goes along the computational domain. The layer will be displayed after at least one step of computation is done.
- To obtain this result, the program will require more then 3000 steps of the computation.

4.1.7 Hypersonic flow around sphere

In this example an external hypersonic flow around a sphere is simulated.



Dimensions:

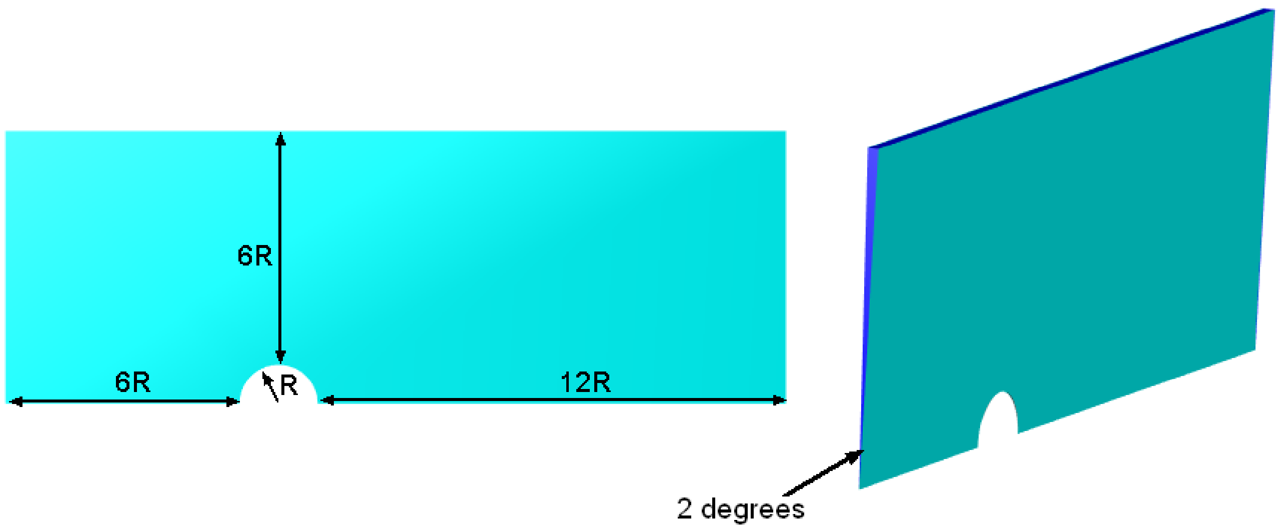
Radius of the sphere:	R	= 0.016	[m]
Substance:		Air	

Flow Settings:

Static pressure:	P	= 100000	[Pa]
Temperature at infinity:	T	= 273	[K]
Velocity at infinity:	V	=3830	[m s^{-1}]
Mach number at infinity:	M	= 11.2	

Geometry: **`Sphere.STL`**
Project: **`Sphere`**
Note:
Computation of this project may require long time and significant amount of computing resources.

4.1.7.1 Computational domain



This problem require simulation of an axisymmetric external hypersonic flow. Therefore, you have to create a computational domain, which is a region with flow around a streamlined body, which is a sector with a small degree (eg 1-2 degrees). When the gas flows at a velocity corresponding to the Mach number > 0.1 it is possible to use non-reflecting boundary conditions. They make less disturbance to the flow, so the distance from the body to the limits of the region, in many cases, can be set less than 10 × characteristic body size.

You can find fully prepared geometry of the computational domain is in the file **`Sphere.STL`**.

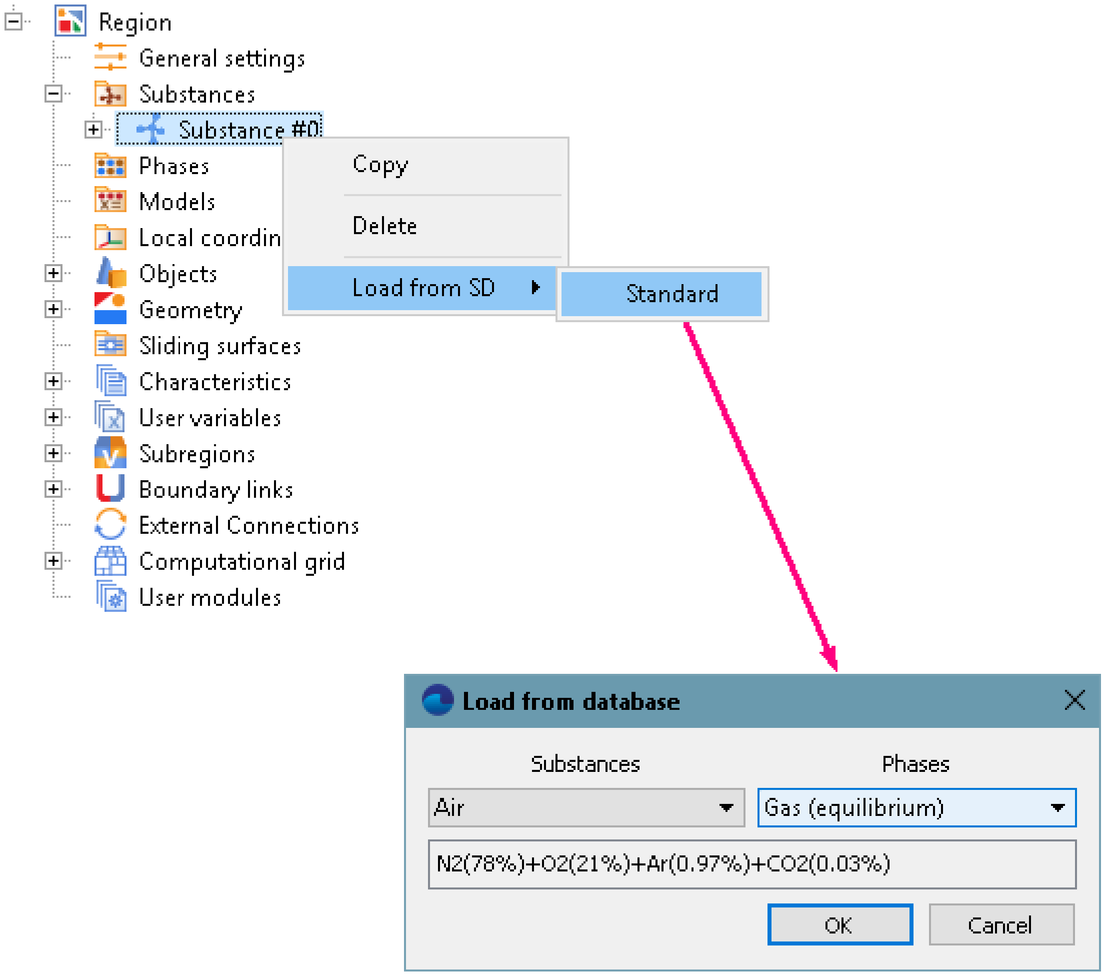
4.1.7.2 Physical model

Do the following steps in the project tree in the **Preprocessor** tab.

In the folder **Substances**:

- Create **Substance #0**.
- Load **Substance #0** from the **Standard** substance database:
  - From the context menu of **Substance #0** select **Load from SD > Standard**.
  - In the new window **Load from database**, select:

Substances	**= Air**
Phases	**= Gas (equilibrium)**



In the folder **Phases**:

- Create a continuous **Phase #0**.
- In **Phase #0** add **Air_Gas (equilibrium)** substance into the folder **Phase #0 > Substances**.
- Specify properties of **Physical processes**:

Motion	= **Navier-Stokes model**
Heat transfer	= **Heat transfer via H**

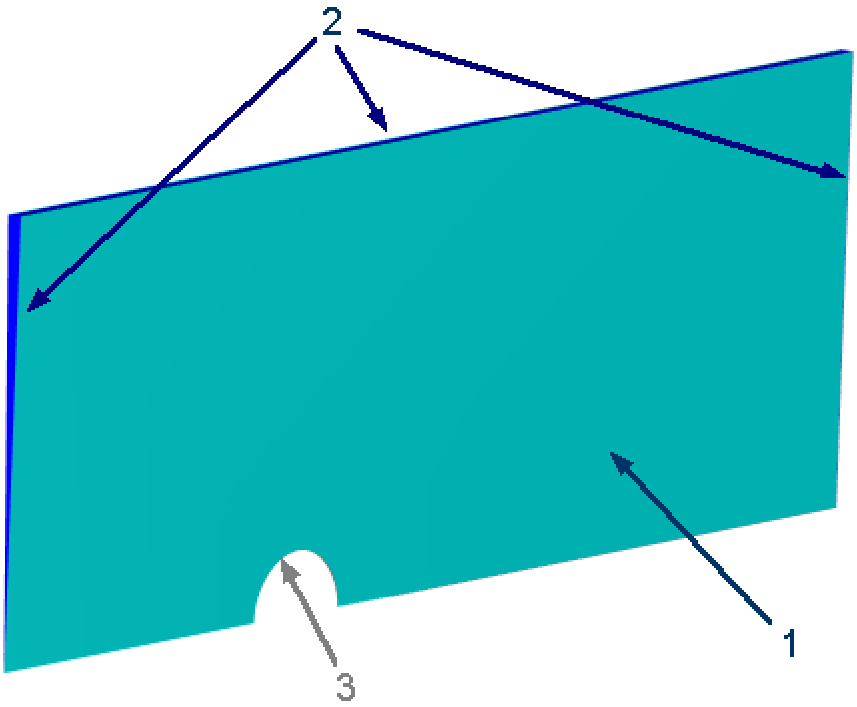
In the folder **Models**:

- Create **Model #0**.
- Add **Phase #0** into subfolder **Model #0 > Phases**

4.1.7.3 Boundary conditions

In the **Properties** window of the **SubRegion #0**, specify:

Model = Model #0



Specify the following boundary conditions:

Boundary 1

Type	**= Symmetry**	
Variables		
Temperature(Phase #0)	**= Symmetry**	
Velocity(Phase #0)	**= Slip**	

Boundary 2

Type	**= Non-reflecting**	
Variables		
Temperature(Phase #0)	**= Non-reflect.**	
Value	**= 0**	[K]
Velocity(Phase #0)	**= Non-reflect.**	
Velocity at inf.		[m s^{-1}]
X	**= 3830**	
Y	**= 0**	
Z	**= 0**	
Pressure at inf.	**= 0**	[Pa]

Boundary 3

Type	**= Wall**	
Variables		
Temperature(Phase #0)	**= Zero gradient**	

Velocity(Phase #0) **= No slip**

4.1.7.4 Initial conditions

In supersonic and hypersonic external flow around bluff bodies we recommend to specify near the body initial values of variables, corresponding to the parameters of flow deceleration, and in the rest of the domain specify initial conditions, which are relevant to parameters of the undisturbed flow.

To set initial conditions, which are relevant to parameters of the undisturbed flow, in properties of the element **Model #0 > Init. data > Init. data #0** specify:

Velocity(Phase #0)

X	**= 3830**	[m s^{-1}]

To set initial conditions corresponding to the deceleration of the flow near the body, you have to:

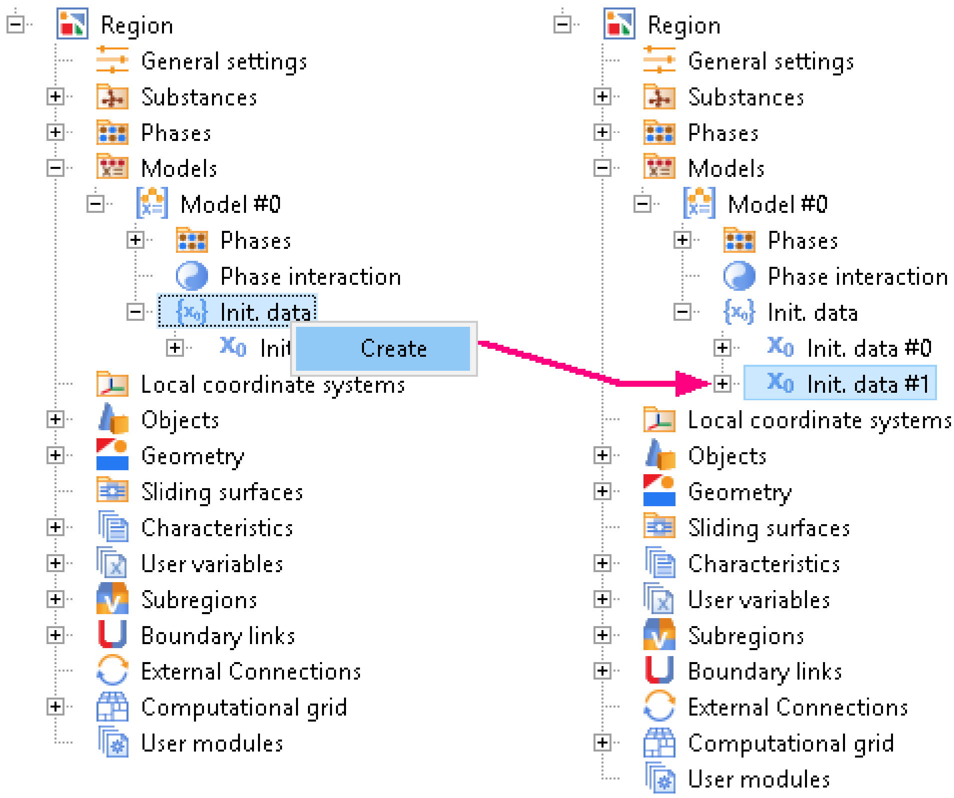
1. Specify **Initial data** corresponding to the parameters of flow deceleration.
2. Specify an **Object** corresponding to the region around the sphere.
3. Specify **Initial conditions**, establishing correspondence between the **Object** and the **Initial data**.

Specify the **Initial data**:

- From the context menu of the folder **Model #0 > Init. data**, select **Create**.
- In properties of the new just created element **Init. data #1** specify:

Temperature (Phase #0)	**= 7300**	[K]
Pressure (Phase #0)	**= 1.65e7** *)	[Pa]

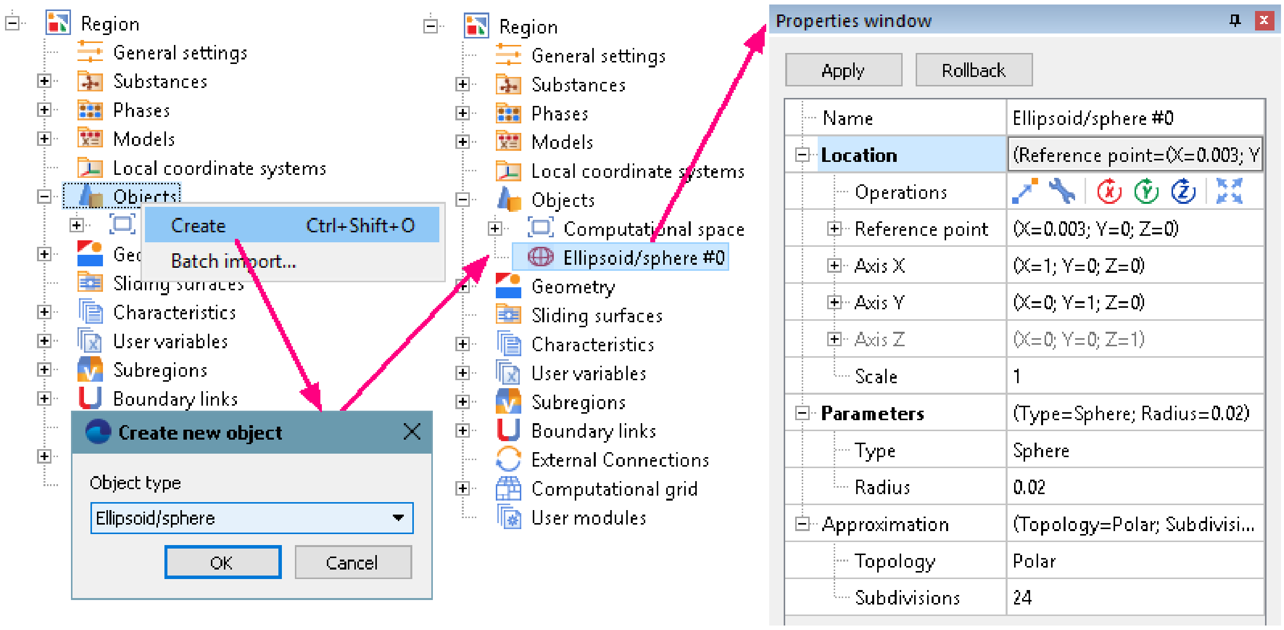
*) **1.65e7** is notation for $1.65x10^7$.



Specify the **Object**:

- From the context menu of the folder **Objects**, select **Create**.

- In the dialog box **Create new object**, which opens, select **Ellipsoid/sphere**.



- In the **Properties** window of **Ellipsoid/sphere #0** specify:

Location

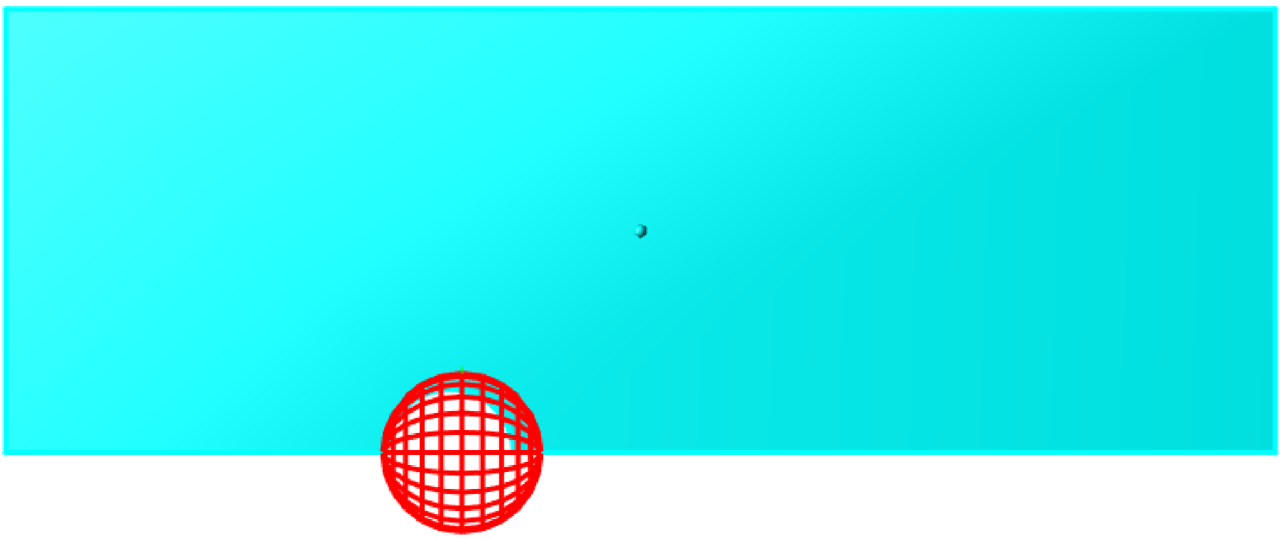
Reference point

X	**= 0.003**	[m]
Y	**= 0**	[m]
Z	**= 0**	[m]

Parameters

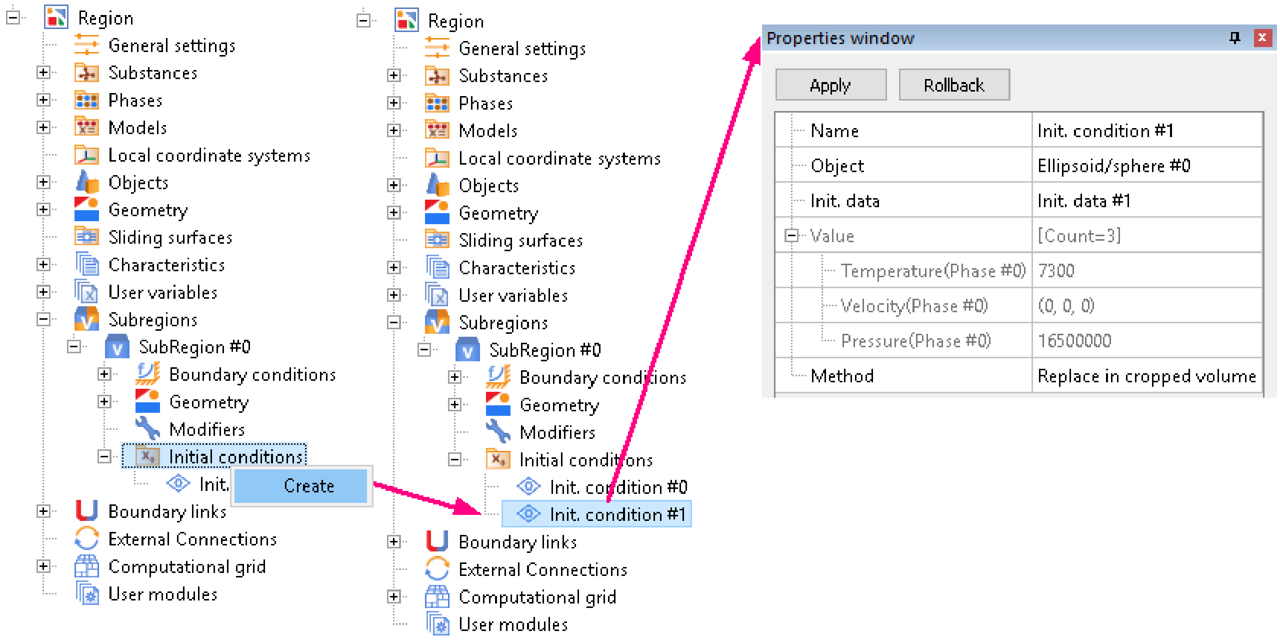
Radius	**= 0.02**	[m]

In the **View** window an image of **Ellipsoid/sphere #0** will appear:



To define a correspondence between the **Object** and the **Initial data**:

- In **SubRegion #0**, from the context menu of the folder **Initial conditions**, select **Create**:



- In the **Properties** window of **Init. condition #1** specify:

Object	**= Ellipsoid/sphere #0**
Init. data	**= Init. data #1**

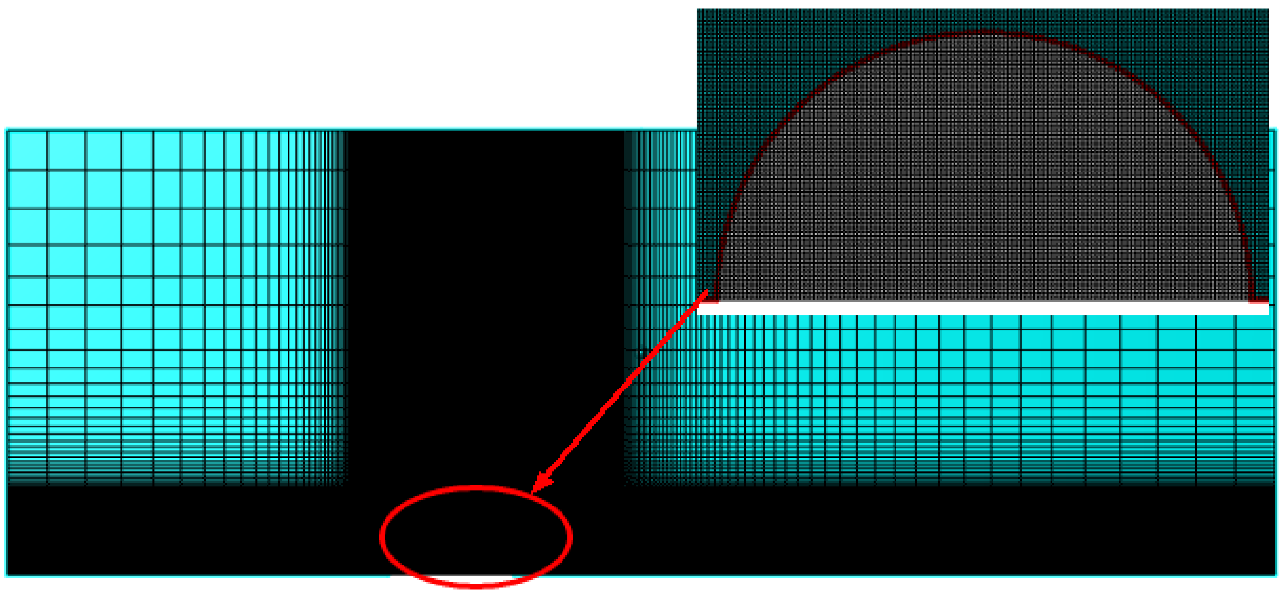
The initial conditions are applied sequentially in the order they appear in the folder **Initial conditions**. Priority of an **Initial condition** is higher when the item is lower in the list[1)]. Thus, in the set-theoretic intersection of **Ellipsoid/spheres #0** and **SubRegion #0**, in the initial moment time parameters of flow deceleration are set, and parameters of the undisturbed flow are set in the rest of the domain .

Note:

[1)] You can change priority of an **Init. conditions** element moving it in the list up and down using the context menu commands **Move Up** and **Move Down**.

4.1.7.5 Initial grid

To better solving the flow near the sphere in this example, specify a two-dimensional non-uniform computational grid, condensed near the sphere.



In the **Properties** window, click the **Initial grid** button to call the **Initial grid editor**.

In the **Initial grid editor** specify:

for axis OX

Grid parameters

h_max = **0.01** [m]

h_min = **0.0002** [m]

Insert a reference line with coordinate **x=0** [m].

Specify **Reference line parameters** for the reference line with coordinate **x=-0.112**:

h = **0.01** [m]

kh+ = **1**

Specify **Reference line parameters** for the reference line with coordinate **x=0**:

h =**0.0002** [m]

kh- = **1.07**

kh+ = **0.97**

Specify **Reference line parameters** for the reference line with coordinate **x=0.208**:

h = **0.01** [m]

kh- = **1**

for axis OY (click the button )

Grid parameters

h_max = **0.01** [m]

h_min = **0.0002** [m]

Specify **Reference line parameters** for the reference line with coordinate **y=0**:

h =**0.0002** [m]

kh+ = **0.95**

Specify **Reference line parameters** for the reference line with coordinate **y=0.112**:

h = **0.01** [m]

kh- = **1**

Click **OK** to close the **Initial grid editor** with saving the entered data.

In properties of the **Initial grid** specify:

Grid structure = 2D

Plane = XY

In the **Properties** window of the **Initial grid** click **Apply**.

4.1.7.6 Parameters of calculation

Specify in the **Solver** tab:

- In properties of the **Time step** element specify:

Method = **Via CFL number**

Convective CFL = **1**

Max step = **0.0001** [s]

- In the **Properties** window of the **Limiters > Limiters for calculation > Phase limiters > Phase #0** element specify:

Limiter

Density, min. = **0.001** [kg m^{-3}]

Temperature abs, min. = **10** [K]

Temperature abs, max. = **1e+5** **) [K]

Velocity, max.	**= 1e+5** **)	[m s^{-1}]
Pressure abs, min.	**= 100**	[Pa]
Pressure abs, max.	**= 1e+8** *)	[Pa]

*) **1e+8** is notation for 10^8.

) **1e+5 is notation for 10^5.

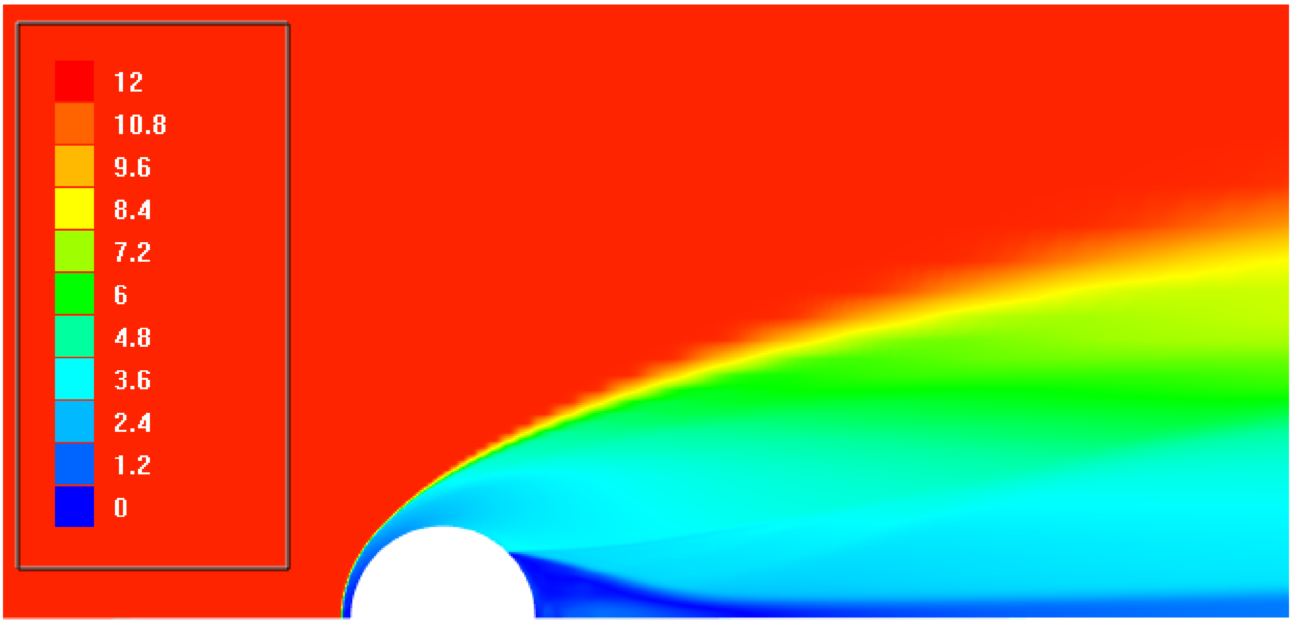
4.1.7.7 Visualization

To view the dynamics of the solution during the computation, prior to computation, specify visualization of:

1. Mach Number distribution in the plane of the flow.
2. Pressure distribution in the plane of the flow.

4.1.7.7.1 Mach Number distribution

Visualization of the Mach Number distribution when the step number is near 7000:



- In the **Properties** window of **Plane #0** specify:

Object

Normal		
	X	**= 0**
	Y	**= 0**
	Z	**= 1**

(to direct a **Plane**'s normal along the axis **Z**, you can also click the **Operations >**  button in the **Properties** window of the **Plane**)

- Create a layer **Color contours** on the **Plane #0**.
- In the **Properties** window of the layer specify:

Variable

Variable	**= Mach number**

Value range

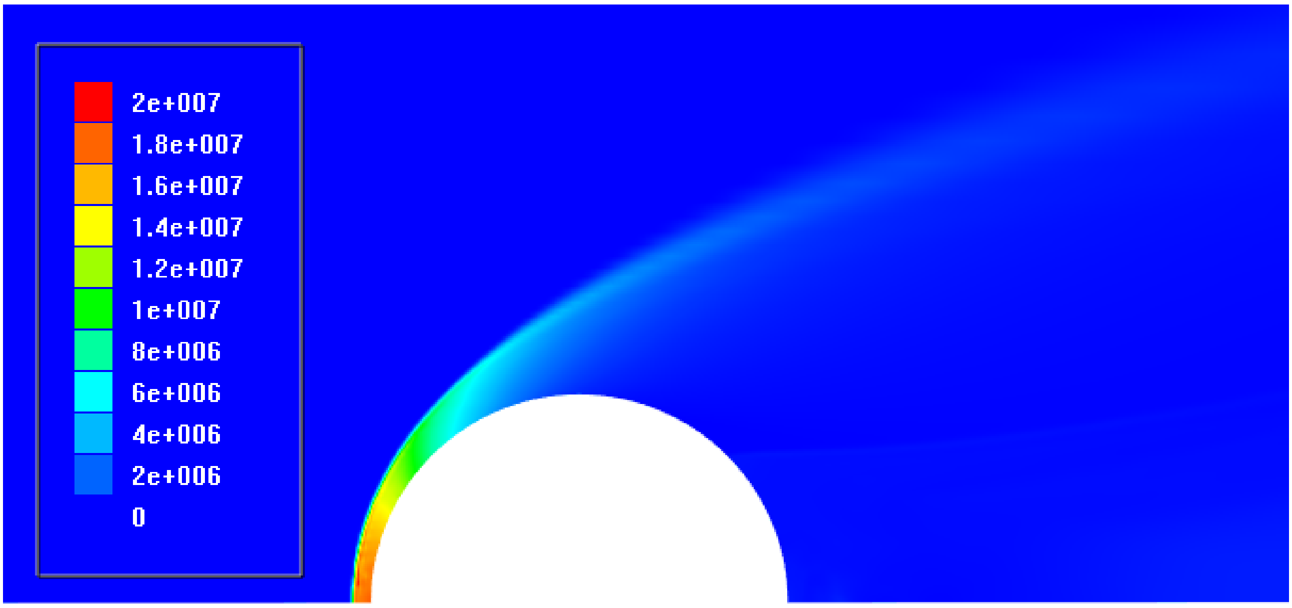
Mode	**= Manual**
Max	**= 12**
Min	**= 0**

Palette

Appearance

Enabled = **Yes**

4.1.7.7.2 Pressure distribution



- Create a layer **Color contours** on **Plane #0**.
- In the **Properties** window of the layer specify:

Variable

Variable = **Pressure**

Value range

Mode = **Manual**

Max = **2e7**

Min = **0**

Palette

Appearance

Enabled = **Yes**

4.2 Heat transfer

In *FlowVision* a convection-diffusion model for heat transfer is implemented.

In order to simulate the convection-diffusion heat transfer, you have to:

- In properties of the **Substance** it is required to specify values of **Density** and **Specific heat** capacity, and, if necessary, the value of the **Thermal conductivity**.
- Enable calculation of the equation of **Heat transfer**.
- Specify the appropriate initial and boundary conditions for the **Temperature**.

4.2.1 Heat transfer in a solid body

Heat transfer in a solid body by means of conduction is considered in the given example.

Parameters of the problem setting

Dimensions:

	The length of the bar	l	= 1	[m]

Parameters of the substance

	Density	ρ	= 7900	[kg m^{-3}]
	Thermal conductivity	λ	= 46	[W $m^{-1}K^{-1}$]
	Specific heat	c_p	= 457	[J kg^{-1} K^{-1}]

Inlet parameters:

	Temperature on the hot wall	T_h	= 100	[K]
	Heat flux from the cold wall	q_c	= -4600	[W m^{-2}]

Geometry: `Conduct.STL`

Project: `Conduct`

4.2.1.1 Physical model

Do the following steps in the project tree in the **Preprocessor** tab.

In the folder **Substances**:

- Create **Substance #0**.
- In properties of **Substance #0** specify:

Aggregative state = **Solid**

Molar mass

Value = **0.056** [kg $mole^{-1}$]

Density

Value	**= 7900**	[kg m^{-3}]
Thermal conductivity		
Value	**= 46**	[W $m^{-1}K^{-1}$]
Specific heat		
Value	**= 457**	[J kg^{-1} K^{-1}]

In the folder **Phases**:

- Create a continuous **Phase #0**.
- In the folder **Phase #0 > Substances** load **Substance #0**.
- In the **Properties** window of the folder **Phase #0 > Physical processes** specify:

Heat transfer = **Heat transfer via h**

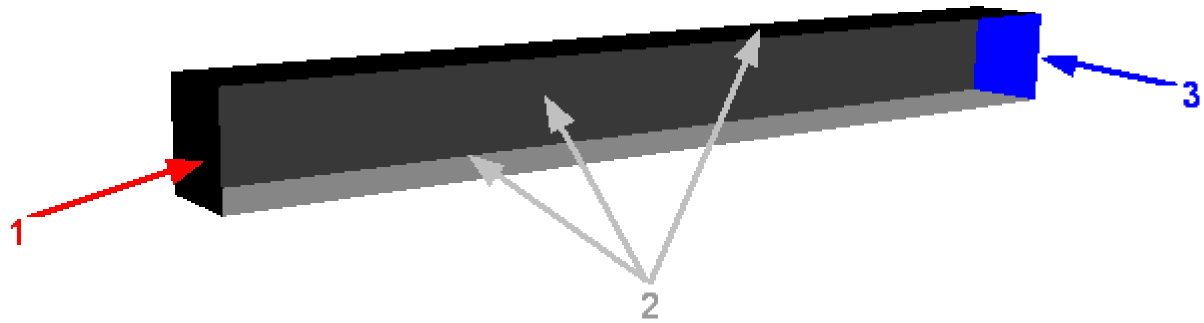
In the folder **Models**:

- Create **Model #0**.
- Add **Phase #0** into subfolder **Model #0 > Phases**

4.2.1.2 Boundary conditions

In the **Properties** window of the **SubRegion #0**, specify:

Model = Model #0



Specify the following boundary conditions:

Boundary 1

Type	**= Wall**	
Variables		
Temperature(Phase #0)	**= Temperature**	
Value	**= 100**	[K]

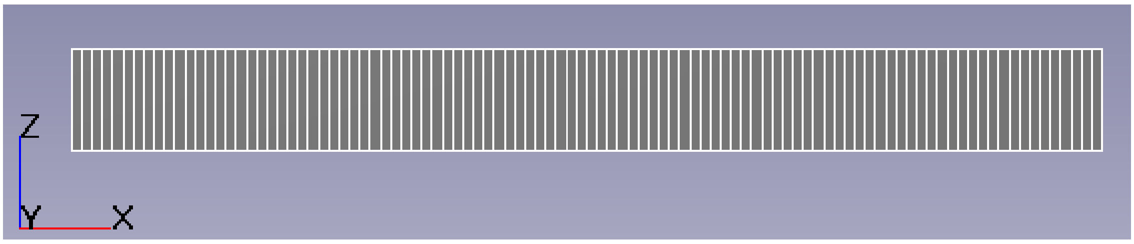
Boundary 2

Type	**= Wall**
Variables	
Temperature(Phase #0)	= **Zero gradient**

Boundary 3

Type	**= Wall**	
Variables		
Temperature(Phase #0)	**= Flux**	
Value	**= -4600**	[W m^{-2}]

4.2.1.3 Initial grid



Specify in the **Properties** window of the **Initial grid**:

Grid structure = 1D

Direction = X

nX = 100

If we used adaptation of the computational grid in this exercise, then adaptation along the non-computational directions **Y** and **Z** would not be done, and the grid would always have only one cell along these directions.

As no adaptation is used here, you can just set **nY=1** and **nZ=1**.

In the **Properties** window of the **Initial grid** click **Apply**.

4.2.1.4 Parameters of calculation

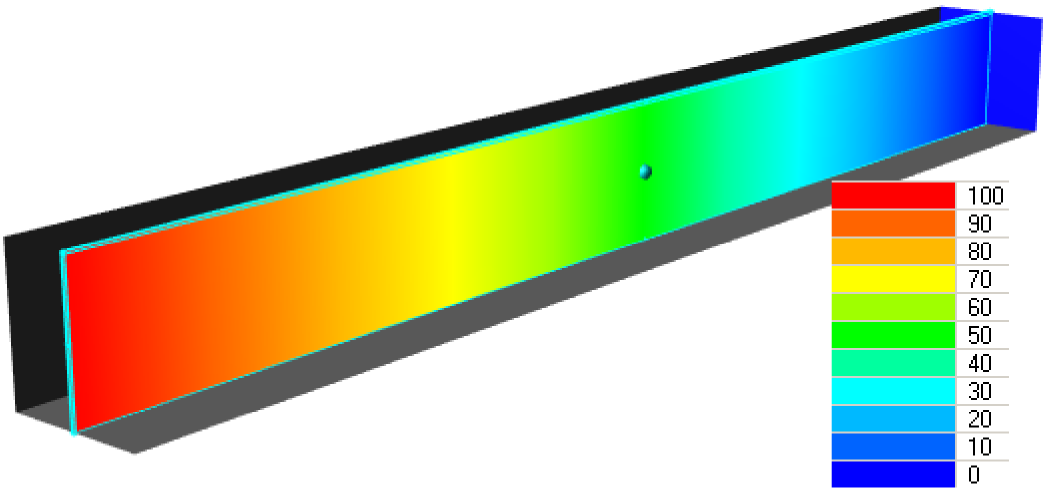
Specify in the **Solver** tab in properties of the **Time step** element:

Method	**= In seconds**	
Constant step	**= 10000**	[s]

4.2.1.5 Visualization

To view the dynamics of the solution during the computation, specify visualization of [Temperature distribution](#) along the length of the bar before the start of computation.

4.2.1.5.1 Temperature distribution



- In the **Properties** window of **Plane #0** specify:

Object

Normal

X	**= 0**
Y	**= 0**
Z	**= 1**

(to direct a **Plane**'s normal along the axis **Z**, you can also click the **Operations >** button in the **Properties** window of the **Plane**)

- Create a layer **Color contours** on the **Plane**.
- In the **Properties** window of the layer specify:

Variable

Variable	**= Temperature**

Value range

Mode	**= Manual**
Max	**= 100**
Min	**= 0**

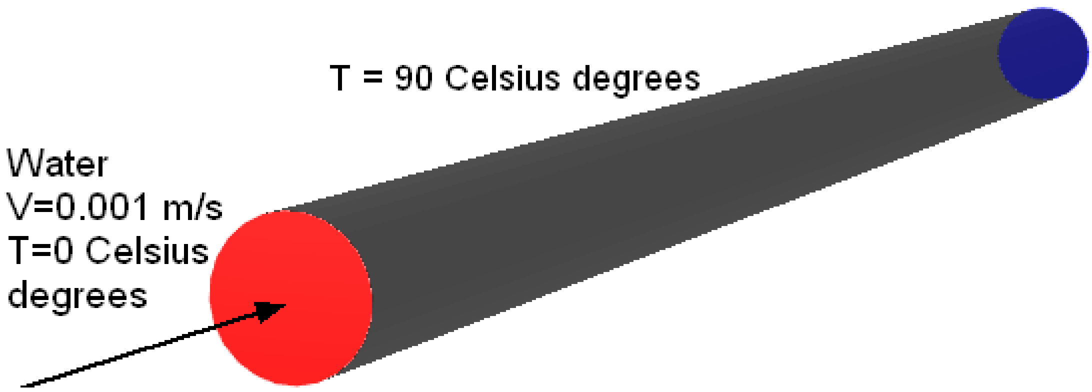
Palette

Appearance

Enabled	**= Yes**
Horiz. alignment	**= Right**
Vert. alignment	**= Bottom**

4.2.2 Forced convection

Consider the simulation of laminar flow of cold water in a tube with a hot wall.



Parameters of the problem setting

Dimensions:

Length of the tube	L	= 2	[m]
Diameter of the tube	D	= 0.1	[m]

Inflow parameters:

Velocity on inlet:	V_{inl}	= 0.001	[m s^{-1}]
Temperature on inlet	T_{inl}	= 0	[°C]
The wall temperature	T_w	= 90	[°C]

Fluid parameters:

Density	ρ	= 1000	[kg m^{-3}]
Viscosity	μ	= 10^{-3}	[kg $m^{-1}s^{-1}$]
Thermal conductivity	λ	= 0.6	[W $m^{-1}K^{-1}$]
Specific heat	c_p	= 4217	[J kg^{-1} K^{-1}]

Reynolds number: $$Re = \frac{V_{inl} D \rho}{\mu} = \frac{0.001 \cdot 0.1 \cdot 1000}{0.001} = 10^2$$

Geometry: `Tube.wrl`

Project: `ForceConvection`

4.2.2.1 Physical model

Do the following steps in the project tree in the **Preprocessor** tab.

In the folder **Substances**:

- Create **Substance #0**.
- In properties of **Substance #0** specify:

Aggregative state	**= Liquid**	
Molar mass		
Value	**=0.018**	[kg mole^{-1}]
Density		
Value	**= 1000**	[kg m^{-3}]
Viscosity		
Value	**= 0.001**	[kg m^{-1}s^{-1}]
Thermal conductivity		
Value	**= 0.6**	[W m^{-1}K^{-1}]
Specific heat		
Value	**= 4217**	[J kg^{-1} K^{-1}]

In the folder **Phases**:

- Create a continuous **Phase #0**.
- In **Phase #0** add **Substance #0** into the folder **Substances**.
- Specify properties of **Phase #0 > Physical processes**:

Heat transfer	= **Heat transfer via h**
Motion	**= Navier-Stokes model**

In the folder **Models**:

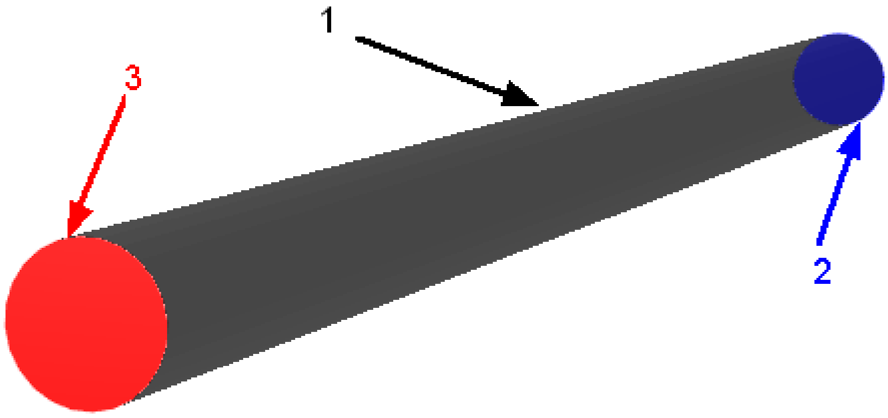
- Create **Model #0**.
- Add **Phase #0** into subfolder **Model #0 > Phases**
- In the folder **Init. data #0** specify:

Velocity(Phase #0)		
X	**= 0**	[m s^{-1}]
Y	**= 0**	[m s^{-1}]
Z	**= 0.001**	[m s^{-1}]

4.2.2.2 Boundary conditions

In the **Properties** window of the **SubRegion #0**, specify:

Model = Model #0



Specify the following boundary conditions:

Boundary 1

Type	**= Wall**	
Variables		
Temperature(Phase #0)	**= Temperature**	
Value	**= 90**	[K]
Velocity(Phase #0)	**= No slip**	

Boundary 2

Type	**= Free Outlet**	
Variables		
Temperature(Phase #0)	**= Zero gradient**	
Velocity(Phase #0)	**= Pressure**	
Value	**= 0**	[Pa]

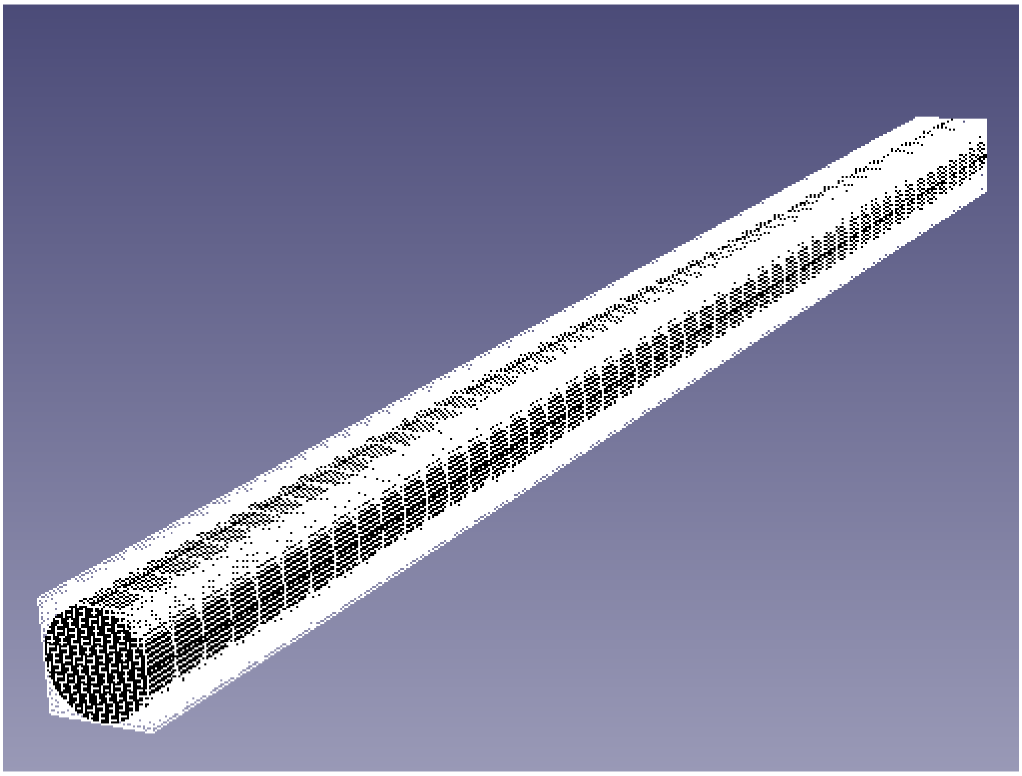
Boundary 3

Type	**= Inlet/Outlet**	
Variables		
Temperature(Phase #0)	**= Temperature**	
Value	**= 0**	[K]
Velocity(Phase #0)	**= Normal mass velocity**	

Specify the numerical value of the **Mass velocity**:

Velocity (Phase #0)		
Mass velocity	= **1**	[kg m^{-2} s^{-1}]

4.2.2.3 Initial grid



Specify in the **Properties** window of the **Initial grid**:

nX	**= 20**
nY	**= 20**
nZ	**= 50**

In the **Properties** window of the **Initial grid** click **Apply**.

4.2.2.4 Parameters of calculation

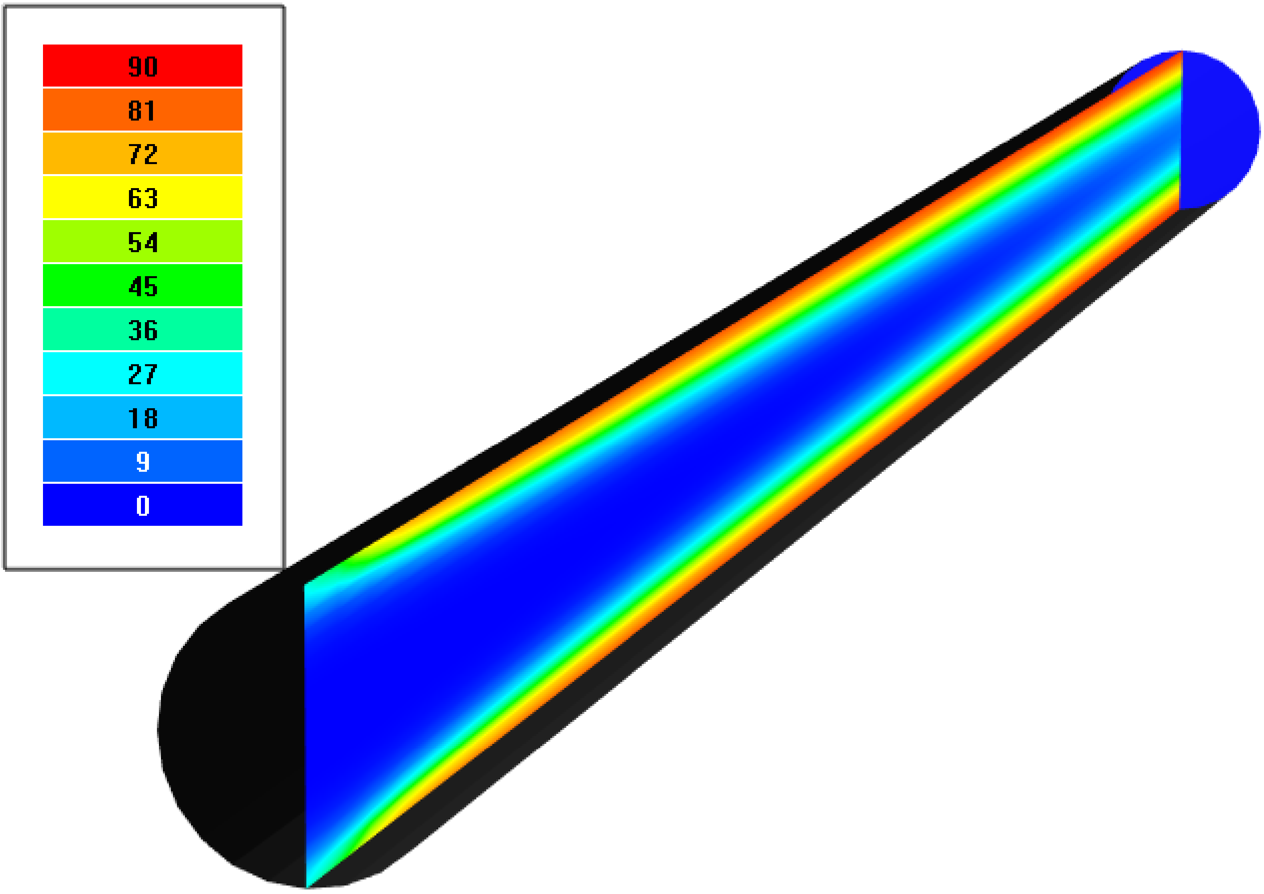
Specify in the **Solver** tab in properties of the **Time step** element:

Method	**= In seconds**	
Constant step	**= 20**	[s]

4.2.2.5 Visualization

To view the dynamics of the solution during the computation, specify visualization of **temperature distribution** in the plane of the flow before the start of computation.

4.2.2.5.1 Temperature distribution

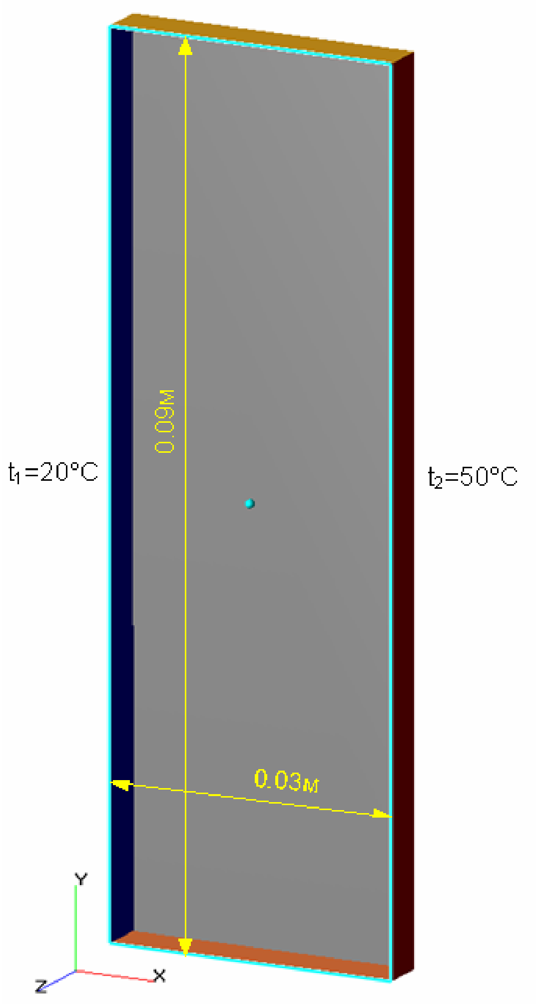


- Create a **Color contours** layer on **Plane #0**.
- In the **Properties** window of the layer specify:

Variable	
Variable	**= Temperature**
Value range	
Mode	**= Manual**
Max	**= 90**
Min	**= 0**
Palette	
Appearance	
Enabled	**= Yes**
Style	**= Style 1**

4.2.3 Natural convection

In this example, the simulation of laminar air flow in a rectangular cavity, vertical walls are maintained at different temperatures (one cold and one hot), while the horizontal walls are insulated. Under the influence of buoyancy due to a temperature difference arises dimensional convective motion.



Parameters of the problem setting

Dimensions of the region:	a × b	= 0.03 × 0.09	[m × m]
Inflow parameters:			
The temperature of the hot wall:	T	= 50	[°C]
The temperature of the cold wall:	T	= 20	[°C]
Parameters of the gas:			
Molar mass	M	= 0.0289	[kg mole^{-1}]
Viscosity	μ	= 1.82×10^{-5}	[kg m^{-1}s^{-1}]
Thermal conductivity	λ	= 0.026	[W (m K)$^{-1}$]
Specific heat	c_p	= 1009	[J kg^{-1} K^{-1}]
Prandtl number:	Pr = 0.71		
Rayleigh number:	Ra = 10^5		
Geometry:	a box that will be created by means of *FlowVision*		
Project:	**`Natur_Convect`**		

4.2.3.1 Creating the computational domain in the "Geometry" tab

The computational domain can be created by means of *FlowVision* without using files created in external CAD software.

In the exercise Time-varying flow in a tube the computational domain was created in the **Preprocessor** tub, while here we will learn how to create the computational domain in the **Geometry** tab.

Create an empty *FlowVision* project using the **File > Create** command from the main menu.

The **New project** dialog box will open; click there the  (**Create empty project**) button. An empty project will be created with the **Geometry** tab opened.

In the **Geometry** tab follow the steps listed below.

- In the folder **Initial geom. models** create **Box #0** (select **Create** from the context menu of the folder **Initial geom. models** and in the **Create new analytical object** dialog box, which opens, select **Object type** = **Box**). Click **OK**.
- In properties of **Box #0** specify:

Object

Location

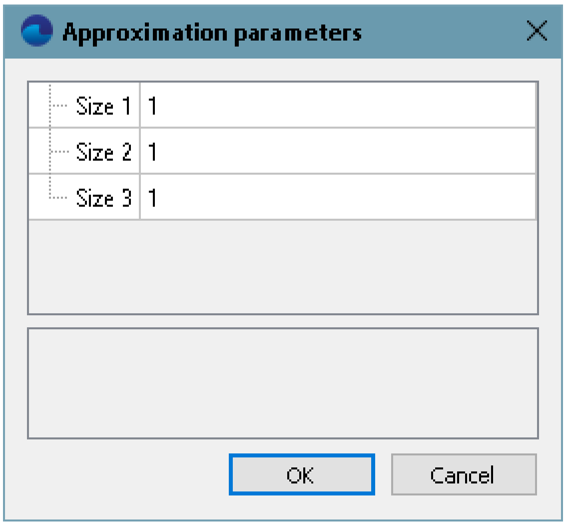
Reference point

X	**= 0.015**	[m]
Y	**= 0.045**	[m]
Z	**= -0.005**	[m]

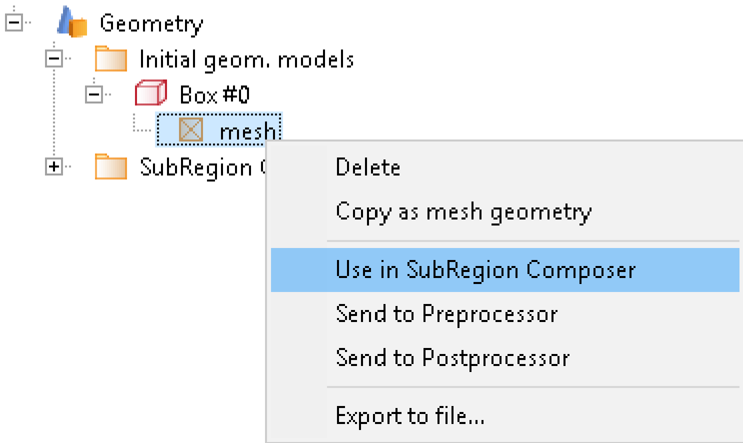
Size

X	**= 0.03**	[m]
Y	**= 0.09**	[m]
Z	**= 0.01**	[m]

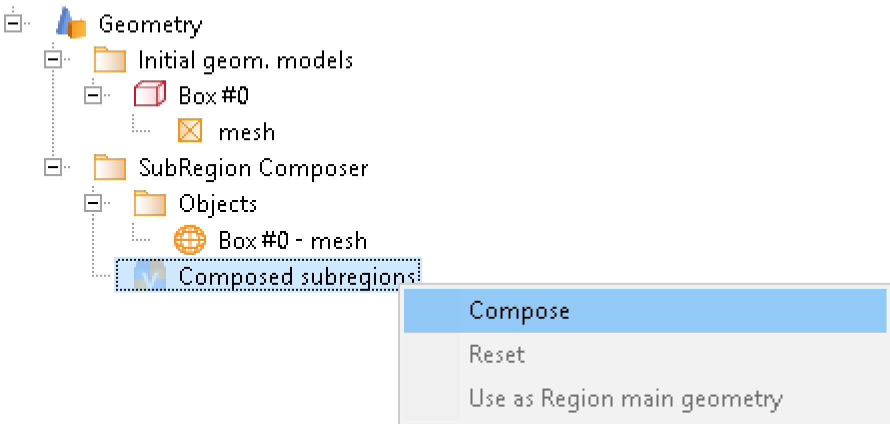
- Click the **Apply** screen button.
- Open the context menu of **Box #0** and select the command **Create consistent mesh**. The **Approximation parameters** dialog box will open where you can specify number of segments along coordinate axes (the more is number of segments, the more precisely the object's geometry will be resolved and the smaller the facets will be). Don't change the default settings and click **OK**:



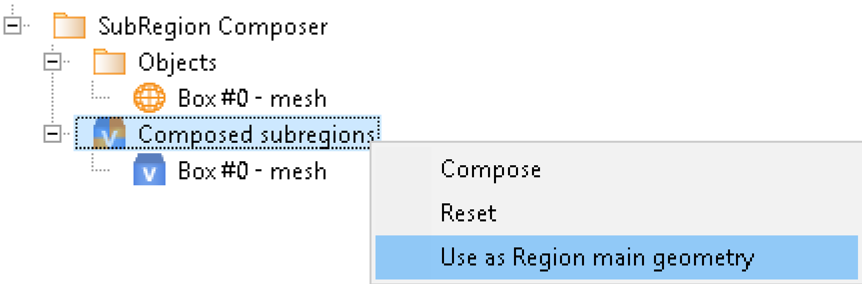
- The **Initial geom. models > Box #0 > mesh** will appear in the project tree. Open its context menu and select the command **Use in SubRegion Composer**:



- The **SubRegion Composer > Objects > Box #0 - mesh** element will appear in the project tree. This element corresponds to the geometry model, which is being created.
- Open the context menu of the element **SubRegion Composer > Composed subregions** (icon of the element is faded now) and select the command **Compose**:



- The **SubRegion Composer > Composed subregions > Box #0 - mesh** element will appear in the project tree.
- Open again the context menu of the element **SubRegion Composer > Composed subregions** and select the command **Use as Region main geometry**:



- The **Preprocessor** tab will open automatically. The just created geometry model of the computational domain will be loaded there.

4.2.3.2 Physical model

In the **Properties** window of the element **General settings** specify:

Reference values

Temperature	**= 293**	[K]

Gravity vector

X	**= 0**	[m s^{-2}]
Y	**= -9.8**	[m s^{-2}]

Z	**= 0**	[m s^{-2}]
g-Density	**= 1.224**	[kg m^{-3}]

In the folder **Substances**:

- Create **Substance #0**.
- Specify properties of **Substance #0**:

Aggregative state = Gas

Properties of child elements in the project tree:

Molar mass			
	Value	**= 0.0289**	[kg $mole^{-1}$]
Density		**= Ideal gas law**	
Viscosity			
	Value	= 1.82×10^{-5}	[kg $m^{-1}s^{-1}$]
Thermal conductivity			
	Value	**= 0.026**	[W $m^{-1}K^{-1}$]
Specific heat			
	Value	**= 1009**	[J kg^{-1} K^{-1}]

In the folder **Phases**:

- Create a continuous **Phase #0**.
- In **Phase #0** add **Substance #0** into the folder **Substances**.
- Specify properties of **Phase #0 > Physical processes**:

Motion	**= Navier-Stokes model**
Heat transfer	**= Heat transfer via h**

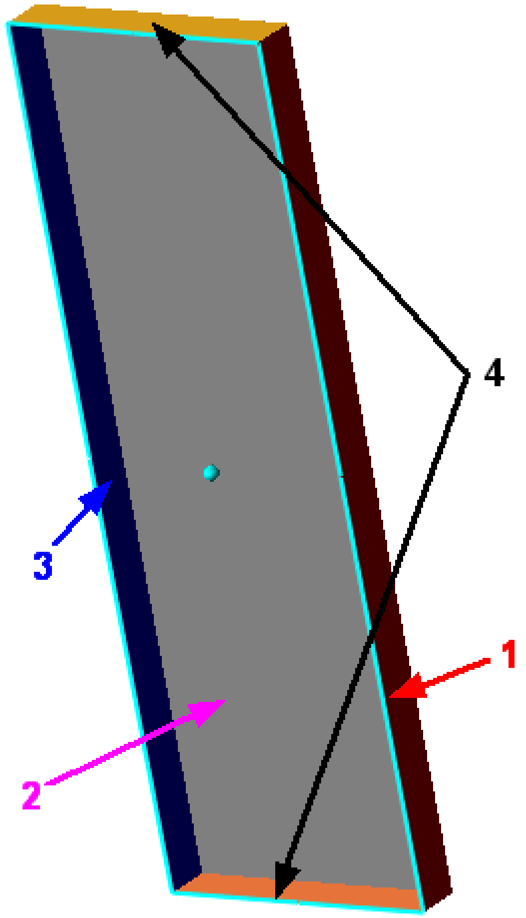
In the folder **Models**:

- Create **Model #0.**
- Add **Phase #0** into subfolder **Model #0 > Phases**.

4.2.3.3 Boundary conditions

In the **Properties** window of the subregion **Box #0 - mesh**, specify:

Model = Model #0



Specify the following boundary conditions:

Boundary 1

Type = **Wall**

Variables

Temperature(Phase #0) = **Temperature**

Value = **30** [K]

Velocity(Phase #0) = **No slip**

Boundary 2

Type = **Symmetry**

Variables

Temperature(Phase #0) = **Symmetry**

Velocity(Phase #0) = **Slip**

Boundary 3

Type = **Wall**

Variables

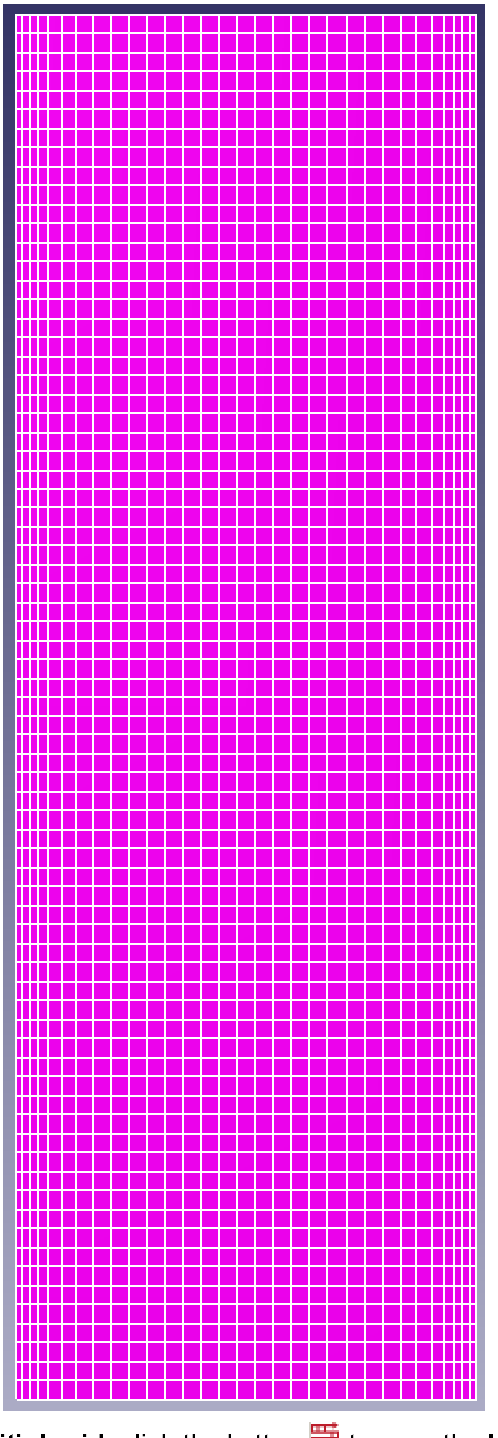
Temperature(Phase #0) = **Temperature**

Value	**= 0**	[K]
Velocity(Phase #0)	**= No slip**	

Boundary 4

Type	**= Wall**
Variables	
Temperature(Phase #0)	**= Zero gradient**
Velocity(Phase #0)	**= No slip**

4.2.3.4 Initial grid



In the **Properties** window of the **Initial grid**, click the button to open the **Initial grid editor**.

Specify in the **Initial grid editor**:

for axis OX

Grid parameters

h_max	**= 0.00125**	[m]
h_min	**= 0.0004**	[m]

Insert reference lines with coordinates:

x1	= 0.005	[m]
x2	= 0.025	[m]

Specify **Reference line parameters** for the reference line with coordinate **x=0**:

h	**= 0.0004**	[m]

Specify **Reference line parameters** for the reference line with coordinate **x=0.005**:

h	**= 0.00125**	[m]

Specify **Reference line parameters** for the reference line with coordinate **x=0.025**:

h	**= 0.00125**	[m]

Specify **Reference line parameters** for the reference line with coordinate **x=0.03**:

h	**= 0.0004**	[m]

Click **OK** to close the **Initial grid editor** with saving the entered data.

Specify in the **Properties** window of the **Initial grid**:

Grid structure	**= 2D**
Plane	**= XY**
nY	**= 73**

In the **Properties** window of the **Initial grid** click **Apply**.

4.2.3.5 Parameters of calculation

Specify in the **Solver** tab in properties of the **Time step** element:

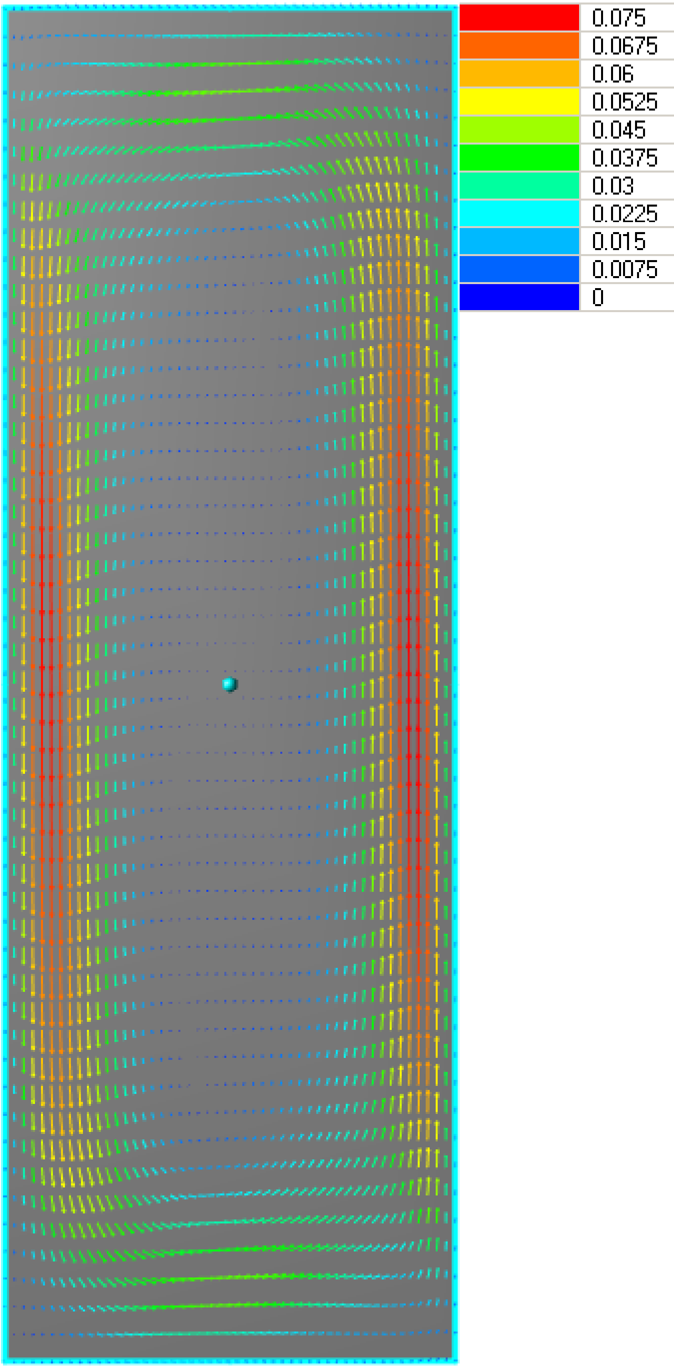
Method	**= In seconds**	
Constant step	**= 0.2**	[s]

4.2.3.6 Visualization

To view the dynamics of the solution during the computation, prior to computation, specify visualization of:

1. **Velocity distribution** in the plane of the flow
2. **Temperature distribution** in the plane of the flow

4.2.3.6.1 Velocity distribution



- In the **Properties** window of **Plane #0** specify:

  Object

  Reference point

  Z = **-0.005**

  Normal

  X = **0**

  Y = **0**

  Z = **1**

  (to direct a **Plane**'s normal along the axis **Z**, you can also click the **Operations >** button in the **Properties** window of the **Plane**)

- Create a layer **Vectors** on **Plane #0**.

- In the **Properties** window of the layer specify:

Grid

- **Size 1** = 50
- **Size 2** = 50

Coloring

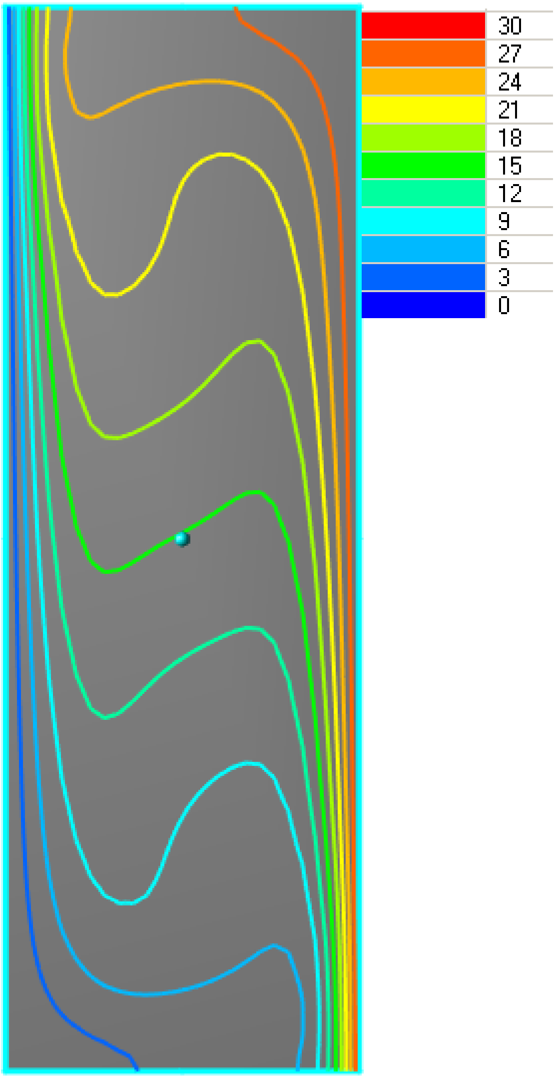
- **Variable**
  - **Variable** = Velocity
- **Value range**
  - **Mode** = Manual
  - **Max** = 0.075
  - **Min** = 0

The program will automatically specify the variable, which is used to build the vectors, **Variable > Variable = Velocity**.

Note:

To display the palette, which is used for coloring the layer (in our case this is coloring the vectors by their absolute values), you can use the **Info** window or, in properties of the layer, set **Appearance > Enabled = Yes**. To open the **Info** window, you have to select the layer in the tree of **Postprocessor** and click the  button.

4.2.3.6.2 Temperature distribution



- Create a layer **Color contours** on **Plane #0**.
- In the **Properties** window of the layer specify:

Variable

Variable	**= Temperature**

Value range

Mode	**= Manual**
Max	**= 30**
Min	**= 0**

Method **= Isolines**

Notes:

- To display the palette, which is used for coloring the layer, you can use the **Info** window or, in properties of the layer, set **Appearance > Enabled = Yes**. To open the **Info** window, you have to select the layer in the tree of **Postprocessor** and click the button.
- If you wish to hide some layers or objects displayed in the **View** window, use the **Hide** command from their context menus.

4.3 Turbulence

FlowVision mainly uses the following turbulence models:

1. **The standard k-ε model**
2. **Low-Reynolds k-ε model AKN**
3. **Quadratic k-ε model**
4. **SST model**
5. **SA model**

In order to simulate the turbulent motion of the liquid or gas:

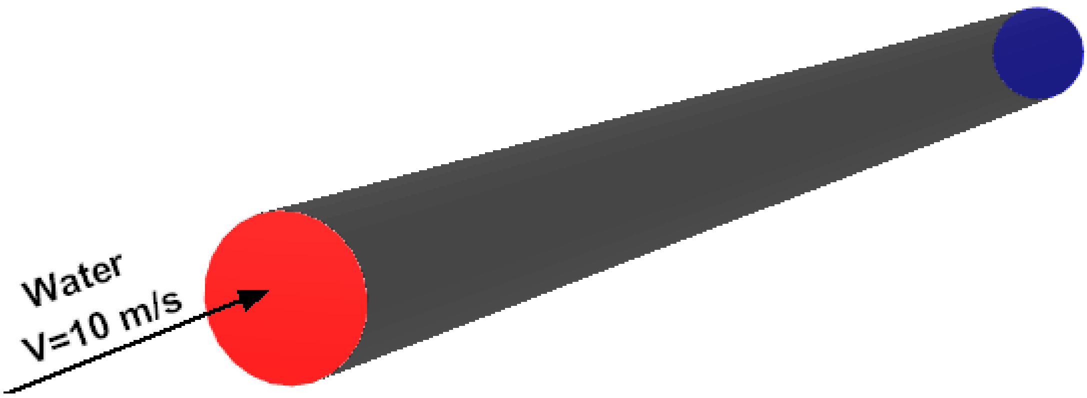
- In a **Substance** specify **Aggregative state** = **Liquid** or **Gas**.
- In properties of the **Substance** specify values of **Density** and **Viscosity**.
- Enable computation of equations of the **Motion** and the corresponding **Turbulence** model.
- Specify the appropriate initial and boundary conditions for the parameters of the turbulence.
- When using the Low-Reynolds k-ε model we recommend you to run a preliminary computation using the standard k-ε model.
- If high gradients of the variables or divergence of the equations of turbulent transport appear in the process of computation, we recommend to set a limiter on the maximum value of the turbulent viscosity, and also specify non-zero background values of turbulent variables in initial conditions and boundary conditions.

4.3.1 Turbulent flow in a tube

The standard k-ε turbulence model is intended for simulation of flows with values of y+ > 3 and small pressure gradients.

Consider an application of the k-ε turbulence model in an example of simulation of turbulent flow in a tube.

A turbulent flow is characterized by the Reynolds number (Re) more then 10^4.



Parameters of the problem setting

Dimensions:

Length of the tube	L	= 2	[m]
Diameter of the tube	D	= 0.1	[m]

Inflow parameters:

Velocity on inlet:	V_{inl}	= 10	[m s^{-1}]

Parameters of the substance:

Density	ρ	= 1000	[kg m^{-3}]
Viscosity	μ	= 0.001	[kg m^{-1}s^{-1}]

Reynolds number: $$\mathrm{Re} = \frac{V_{inl} D \rho}{\mu} = \frac{10 \cdot 0.1 \cdot 1000}{0.001} = 10^6$$

Geometry: `Tube.wrl`

Project: `Tube_turb`

4.3.1.1 Physical model

In the **Properties** window of the element **General settings** specify:

Reference values

Temperature	**= 298**	[K]
Pressure	**= 100000**	[Pa]

In the folder **Substances**:

- Create **Substance #0**.
- Specify properties of **Substance #0**:

Aggregative state	**= Liquid**	
Molar mass		
Value	**=0.018**	[kg mole^{-1}]
Density		
Value	**= 1000**	[kg m^{-3}]
Viscosity		
Value	**= 0.001**	[kg m^{-1}s^{-1}]
Specific heat		
Value	**= 4217**	[J kg^{-1} K^{-1}]

In the folder **Phases**:

- Create a continuous **Phase #0**.
- In **Phase #0** add **Substance #0** into the folder **Substances**.
- Specify properties of the folder **Phase #0 > Physical processes**:

Motion	**= Navier-Stokes model**
Turbulence	**= KES**

In the folder **Models**:

- Create **Model #0**.
- Add **Phase #0** into subfolder **Model #0 > Phases**
- In the folder **Init. data #0** specify:

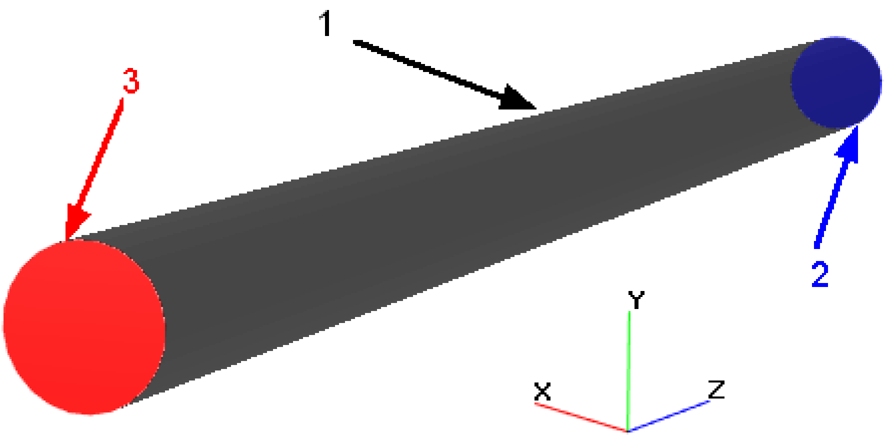
Velocity (Phase #0)

Z	**= 10**	[m s^{-1}]

4.3.1.2 Boundary conditions

In the **Properties** window of the **SubRegion #0**, specify:

Model = Model #0



Specify the following boundary conditions:

Boundary 1

Name	**= Wall**	
Type	**= Wall**	
Variables		
Velocity (Phase #0)	**= Logarithm law**	
TurbEnergy (Phase #0)	= **Value in cell near wall**	
TurbDissipation (Phase #0)	= **Value in cell near wall**	

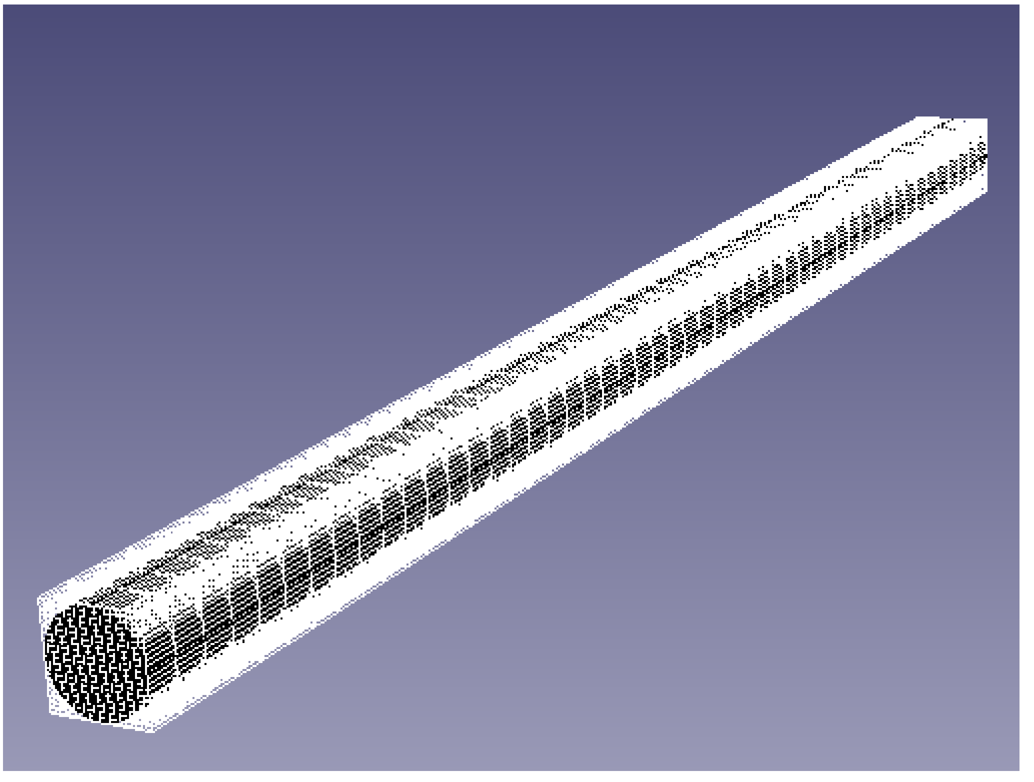
Boundary 2

Name	**= Outlet**	
Type	**= Free Outlet**	
Variables		
Velocity (Phase #0)	**= Pressure**	
Value	**= 0**	[Pa]
TurbEnergy (Phase #0)	**= Zero gradient**	
TurbDissipation (Phase #0)	**= Zero gradient**	

Boundary 3

Name	**= Inlet**	
Type	**= Inlet/Outlet**	
Variables		
Velocity (Phase #0)	**= Normal mass velocity**	
Mass velocity	**= 10000**	[kg m^{-2} s^{-1}]
TurbEnergy (Phase #0)	**= Pulsations**	
Value	**= 0**	
TurbDissipation (Phase #0)	**= Turbulent scale**	
Value	**= 0**	[m]

4.3.1.3 Initial grid



Specify in the **Properties** window of the **Initial grid**:

nX = 20

nY = 20

nZ = 50

In the **Properties** window of the **Initial grid** click **Apply**.

4.3.1.4 Parameters of calculation

Specify in the **Solver** tab in properties of the **Time step** element:

Method	**= In seconds**	
Constant step	**= 0.01**	[s]

4.3.1.5 Visualization

To view the dynamics of the solution during the computation, prior to computation, specify visualization of:

1. **Pressure variation on inlet**
2. **Turbulent viscosity distribution** in the plane of the flow
3. **Velocity distribution** in the plane of the flow
4. **Pressure distribution** along the axis of the tube

4.3.1.5.1 Pressure variation on inlet

We consider below two methods of displaying a plot the pressure variation on inlet. The first steps of these methods are same:

- On the BC **Inlet** create a **Supergroup** in **Preprocessor** (use in the context menu the command **Create supergroup > In Preprocessor**).
- Create **Characteristics** on this **Supergroup**.

- In properties of the created element **Characteristics** > **Characteristics #0 (Supergroup on "Inlet")** specify:

Variable

Variable **= Pressure**

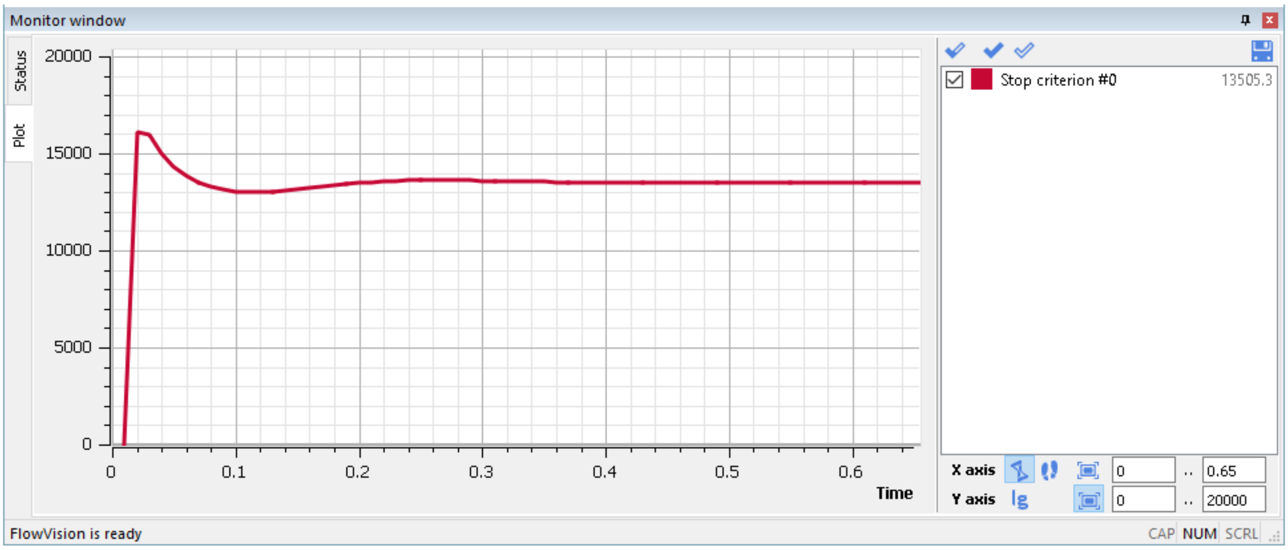
Displaying the plot in the Monitor window

In the **Solver** tab, in the folder **Stopping conditions > User values** create a user **Stop criterion #0** and in its properties specify:

Object **= Characteristics #0 (Supergroup on "Inlet")**

Variable **= <f surf.>**

Run the computation and view dynamics of the pressure variation on inlet in the **Plot** tab of the **Monitor** window (select the **Stop criterion #0** in the right pane and set parameters of the plot if necessary):



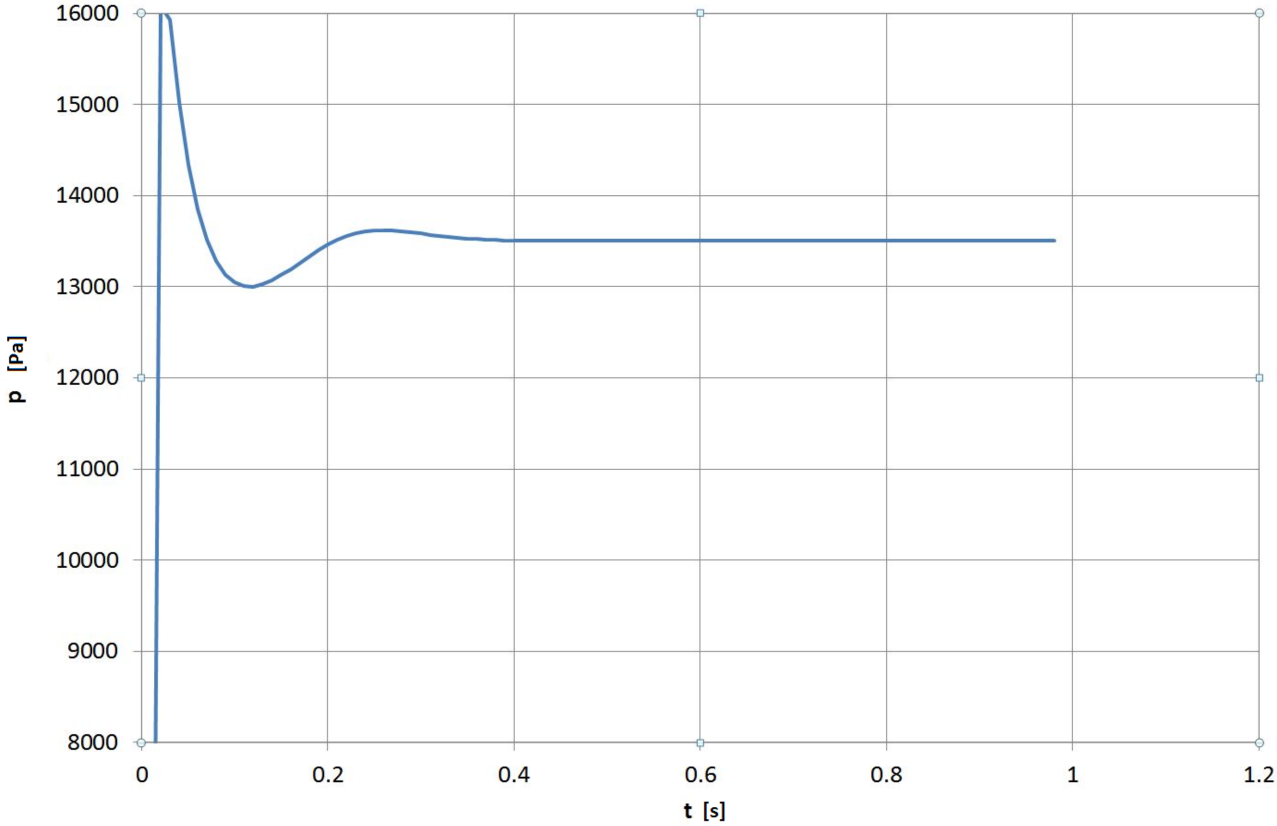
Displaying the plot in an external program

- In the **Postprocessor** tab, in properties of the element **Characteristics** > **Characteristics #0 (Supergroup on "Inlet")**, specify:

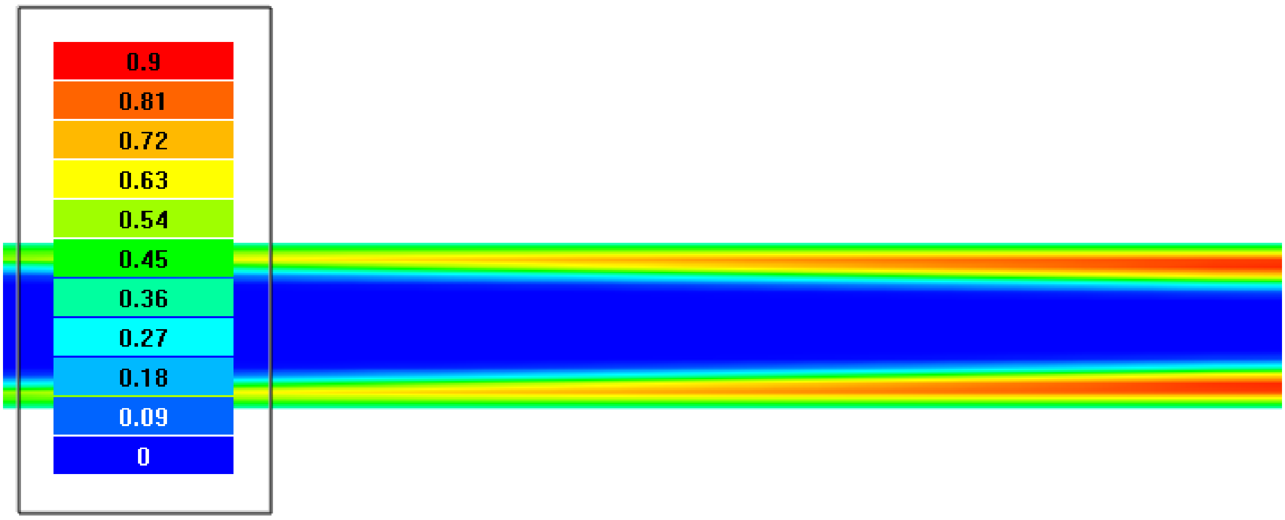
Save to file

Type **= Automatic**

- After the computation is finished, open the `glo`-file, which is specified in properties of the element **Characteristics #0 (Supergroup on "Inlet")**, by an external program (for example, by *Excel*) and plot the dependency of **Avg** by **Time**.



4.3.1.5.2 Turbulent viscosity distribution



- Create a layer **Color contours** on **Plane #0**.
- In the **Properties** window of the layer specify:

Variable	
Variable	**= TurbViscosity**
Value range	
Mode	**= Manual**

Max = 0.9

Min = 0

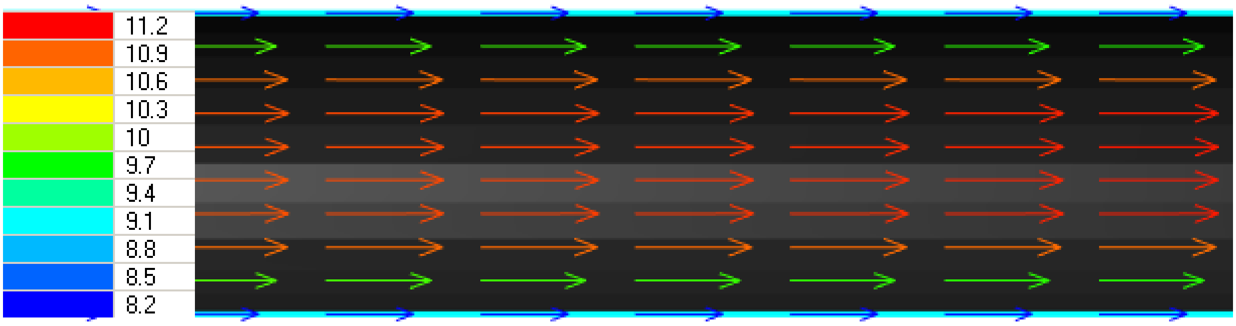
Palette

Appearance

Enabled = Yes

Style = Style 1

4.3.1.5.3 Velocity distribution



- Create a layer **Vectors** on **Plane #0**.
- In the **Properties** window of the layer specify:

Grid

Size 1 = 10

Size 2 = 40

Coloring

Variable

Variable = Velocity

Value range

Mode = Manual

Max = 11.2

Min = 8.2

The program will automatically specify the variable, which is used to build the vectors, **Variable > Variable = Velocity**.

Notes:

- To display the palette, which is used for coloring the layer (in our case this is coloring the vectors by their absolute values), you can use the **Info** window or, in properties of the layer, set **Appearance > Enabled = Yes**. To open the **Info** window, you have to select the layer in the tree of **Postprocessor** and click the button.
- If you wish to hide some layers or objects displayed in the **View** window, use the **Hide** command from their context menus.

4.3.1.5.4 Pressure distribution

- Create **Line #0**.
- In the **Properties** window of **Line #0**, specify:

Object

Reference point

X	**0**
Y	**0**
Z	**0.001**

Direction

X	**0**
Y	**0**
Z	**1**

- Create a layer **Plot along line** on **Line #0**.
- In the **Properties** window of the layer specify:

Variable

Variable **= Pressure**

Value range

Mode **= Manual**

Max **= 13000**

Min **= 0**

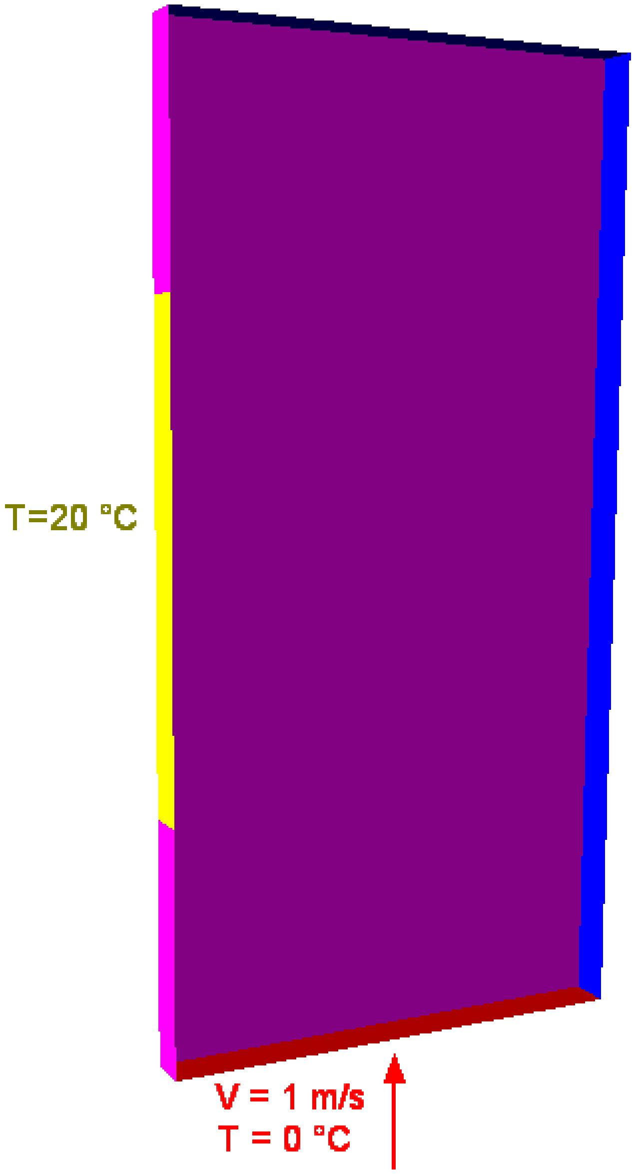
Notes:

- To display information about a layer, open the **Info** window. Select an item in the **Postprocessor** tab of project tree and then click on the *i* button in the toolbar.
- If you wish to hide some layers or objects displayed in the **View** window, use the **Hide** command from their context menus.

4.3.2 Turbulent flow over a plate

The *Low-Reynolds k-ε turbulence model (AKN)* with equilibrium wall functions can be used for simulation of flows with small gradients at y+>3 and with no wall functions at y+<3. In settings without wall functions you have also specify values of parameters of turbulence of the approach flow.

Consider the use of the Low-Reynolds turbulence model without wall functions for simulating the dynamic and the thermal boundary layers on a plate.



Parameters of the problem setting

Dimensions

Length of the computational domain	= 2	[m]
Width of the computational domain	= 1	[m]
Length of the plate	L = 1	[m]

Inflow parameters:

Plate temperature	T = 20	[K]

Velocity on inlet	V = 1	[m s^{-1}]
Temperature on inlet	T = 0	[K]

Parameters of the substance:

Density	ρ= 1000	[kg m^{-3}]
Viscosity	μ= 0.001	[kg m^{-1} s^{-1}]
Thermal conductivity	λ= 0.6	[W m^{-1} K^{-1}]
Specific heat capacity	c_p= 4217	[J kg^{-1} K^{-1}]

Reynolds number $$Re = \frac{VL\rho}{\mu} = \frac{1 \cdot 1 \cdot 1000}{0.001} = 10^6$$

Geometry: **Plate.wrl**

Project: **Plate**

4.3.2.1 Physical model

In the folder **Substances**:

- Create **Substance #0.**
- Specify the following properties of the **Substance #0**:

Aggregative state	**= Liquid**	
Molar mass		
Value	**=0.018**	[kg $mole^{-1}$]
Density		
Value	**= 1000**	[kg m^{-3}]
Viscosity		
Value	**= 0.001**	[kg $m^{-1}s^{-1}$]
Thermal conductivity		
Value	**= 0.6**	[W $m^{-1}K^{-1}$]
Specific heat		
Value	**= 4217**	[J kg^{-1} K^{-1}]

In the folder **Phases**:

- Create a continuous **Phase #0.**
- In **Phase #0** add **Substance #0** into the folder **Substances.**
- Specify properties of the folder **Phase #0 > Physical processes**:

Heat transfer	**= Heat transfer via h**
Motion	**= Navier-Stokes model**
Turbulence	**= KES** *)

In the folder **Models**:

- Create **Model #0.**
- Add **Phase #0** into subfolder **Model #0 > Phases**
- In the folder **Init. data #0** specify:

Velocity (Phase #0)

X	**= 0**	[m s^{-1}]
Y	**= 1**	[m s^{-1}]

Z	**= 0**	[m s^{-1}]
Pulsations (Phase #0)		
Value	**= 0.0001**	
Turbulent scale (Phase #0)		
Value	**= 0.001**	[m]

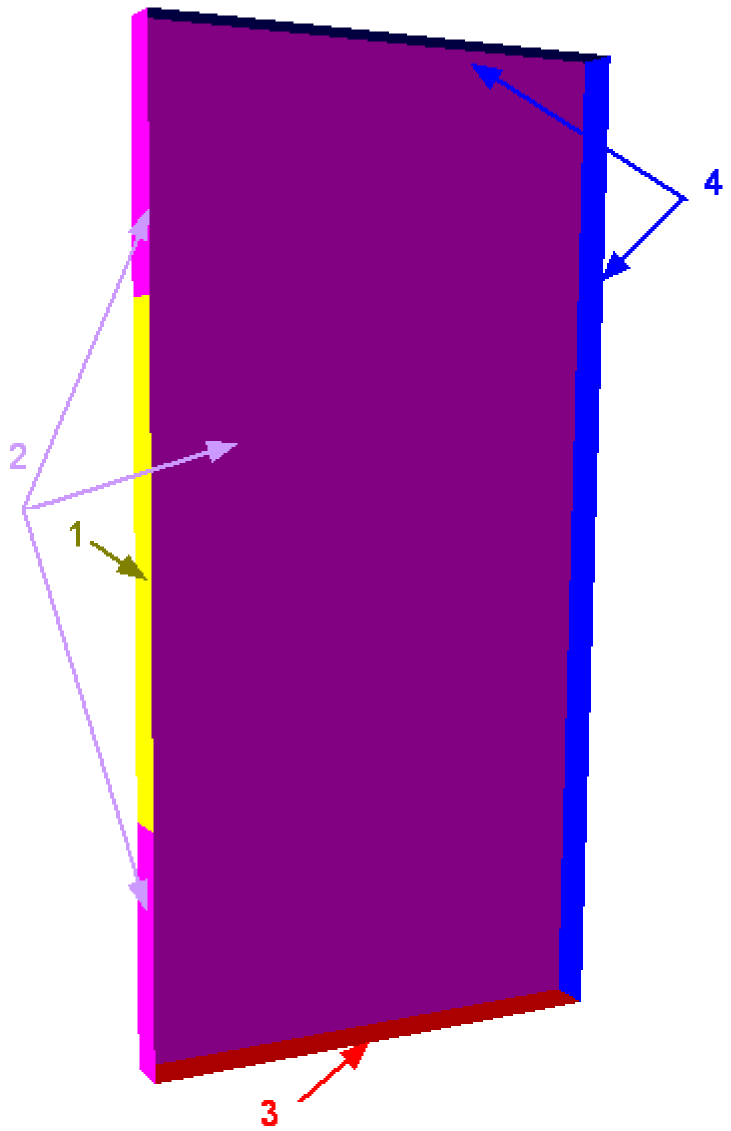
Note:

*) Before applying the *Low-Reynolds k-ε turbulence model (AKN)*, you need to carry out a preliminary computation using the *Standard k-ε turbulence model (KES)* with wall functions.

4.3.2.2 Boundary conditions

In the **Properties** window of the **SubRegion #0**, specify:

Model = Model #0



Specify the following boundary conditions:

Boundary 1

Name	**= Wall**	
Type	**= Wall**	
Variables		
Temperature(Phase #0)	**= Temperature**	
Value	**= 20**	[K]
Velocity(Phase #0)	**= Logarithm law**	
TurbEnergy(Phase #0)	**= Value in cell near wall**	
TurbDissipation(Phase #0)	**= Value in cell near wall**	

Boundary 2

Name	**= Symmetry**	
Type	**= Symmetry**	
Variables		
Temperature(Phase #0)	**= Symmetry**	
Velocity(Phase #0)	**= Slip**	
TurbEnergy(Phase #0)	**= Symmetry**	
TurbDissipation(Phase #0)	**= Symmetry**	

Boundary 3

Name	**= Inlet**	
Type	**= Inlet/Outlet**	
Variables		
Temperature(Phase #0)	**= Temperature**	
Value	**= 0**	[K]
Velocity(Phase #0)	**= Normal mass velocity**	
Mass velocity	**= 1000**	[kg $m^{-2}s^{-1}$]
TurbEnergy(Phase #0)	**= Pulsations**	
Value	**= 0.0001**	
TurbDissipation(Phase #0)	**= Turbulent scale**	
Value	**= 0.001**	[m]

(Here values of values of parameters of turbulence of the approach flow are set because it is planned to do computation using the low-Reynolds k-ε turbulence model (AKN) without wall functions)

Boundary 4

Name	= **Outlet**	
Type	**= Free Outlet**	
Variables		
Temperature(Phase #0)	**= Zero gradient**	
Velocity(Phase #0)	**= Pressure**	
Value	**= 0**	[Pa]
TurbEnergy(Phase #0)	= **Zero gradient**	
TurbDissipation(Phase #0)	= **Zero gradient**	

4.3.2.3 Initial grid

In the **Properties** window of the **Initial grid**, click the button to open the **Initial grid editor**.

Specify in the **Initial grid editor**:

for axis OX

Grid parameters

h_max	**= 0.1**	[m]
h_min	**= 0.0001**	[m]

Specify **Reference line parameters** for the reference line with coordinate **x=0**:

h	**= 0.0001**	[m]
kh+	**= 1**	

Specify **Reference line parameters** for the reference line with coordinate **x=1**:

h	**= 0.1**	[m]
kh-	**= 0.5**	

for axis OY (click the button Y)

Grid parameters

h_max	**= 0.125**	[m]
h_min	**= 0.002**	[m]

Insert reference lines with coordinates:

y1	= -0.5	[m]
y2	= 0.5	[m]

Specify **Reference line parameters** for the reference line with coordinate **y=-1**:

h	**= 0.125**	[m]

Specify **Reference line parameters** for the reference line with coordinate **y=-0.5**:

h	**=0.002**	[m]
kh-	**= 0.9**	
kh+	**= 1.1**	

Specify **Reference line parameters** for the reference line with coordinate **y=0.5**:

h	**=0.002**	[m]
kh-	**= 0.9**	
kh+	**= 1.1**	

Specify **Reference line parameters** for the reference line with coordinate **y=1**:

h	**= 0.125**	[m]

Click **OK** to close the **Initial grid editor** with saving the entered data.

Specify in the **Properties** window of the **Initial grid**:

Grid structure = 2D

Plane = XY

In the **Properties** window of the **Initial grid** click **Apply**.

4.3.2.4 Parameters of calculation

In the **Solver** tab:

In properties of the **Time step** element, specify:

Method	**= In seconds**	
Constant step	**= 0.01**	[s]

In properties of the **Stopping conditions > Time steps** element specify **Number = 100**. This setting will automatically stop the preliminary computation.

4.3.2.5 Preliminary computation

Do a preliminary computation using the *Standard k-ε turbulence model (KES)* of turbulence:

- Run the project on computation.
- When the program makes 100 iterations the computation will stop (as you have specified in parameters of calculation **Number = 100** in properties of the element **Stopping conditions > Time steps**).

Change the turbulence model to *Low-Reynolds k-ε model AKN*.

- In properties of the folder **Phase #0 > Physical processes** specify:

Turbulence	**= KEAKN**

- In properties of the boundary condition **Wall** specify:

Wall interaction

Phase #0	= **No wall functions** *)

- In the **Solver** tab, in properties of the element **Stopping conditions > Time steps**, specify **Number = 1000000** and run the project on computation for continuation.

Notes:

*) When y+ < 3 over most of the surface of the wall, it is recommended to specify the **Wall interaction**= **No wall functions**.

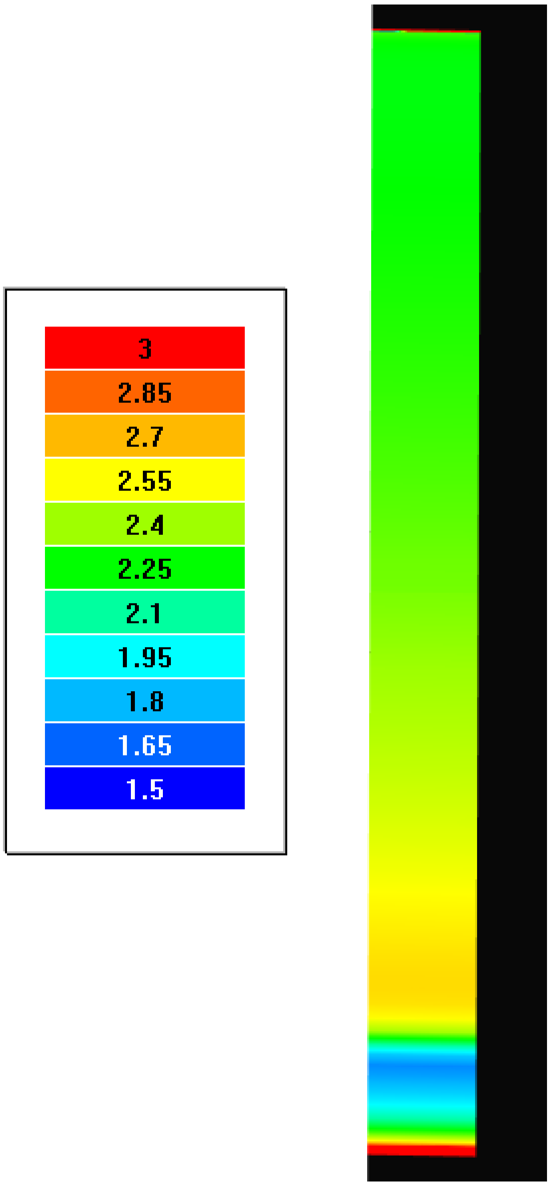
4.3.2.6 Visualization

To view the dynamics of the solution during the computation, prior to computation, specify visualization of:

1. Y+ distribution on a surface of the wall
2. Viscous friction distribution along the plate

4.3.2.6.1 Y+ distribution

Visualization of Y+ distribution when the step number is near 350:



- Create a **Supergroup** on the BC **Wall** using the **Create supergroup > In Postprocessor** command from the context menu.
- Create a layer **Color contours** on this **Supergroup**.

- In the **Properties** window of the layer specify:

Variable

Variable = **Y_plus**

Value range

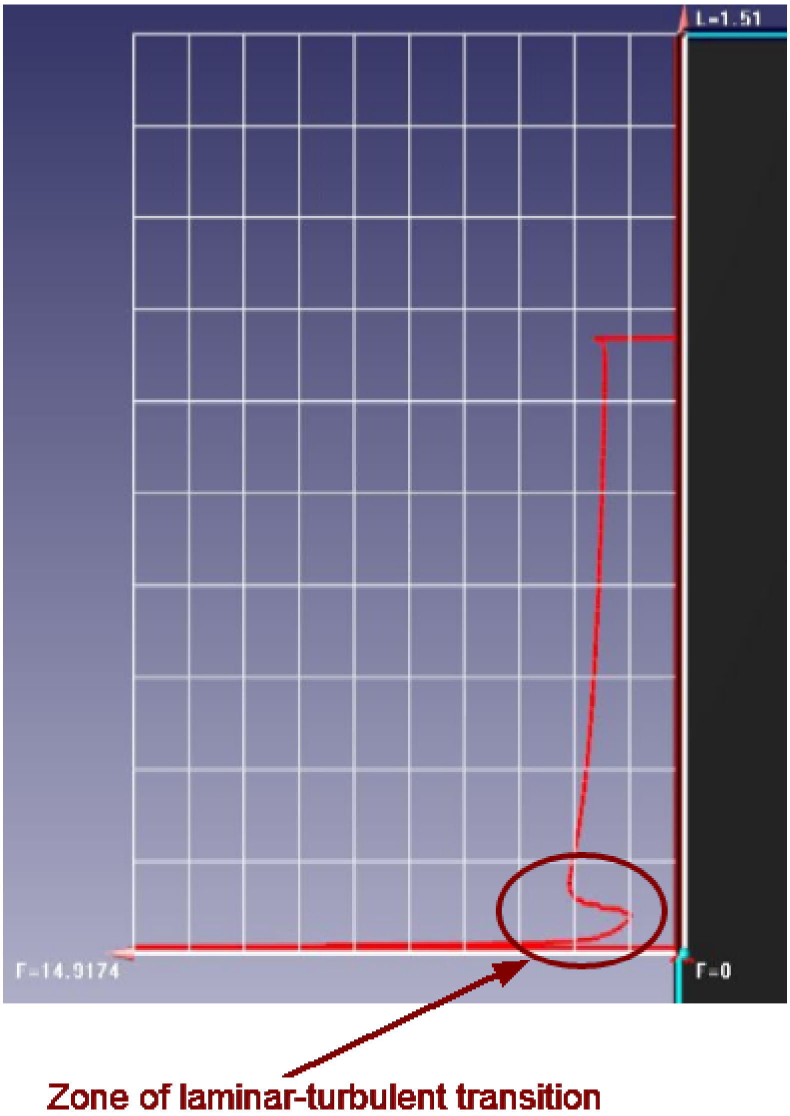
Mode = **Manual**

Max = **3**

Min = **1.5**

4.3.2.6.2 Viscous friction distribution

Visualization of distribution of the viscous friction over the plate when the step number is near 350:



- In the **Properties** window of **Plane #0** specify:

Object

Reference point

X = **0.01**

Y = **-0.51**

Z = **0.05**

Normal

X	**= 0**
Y	**= 0**
Z	**= 1**

(to direct a **Plane**'s normal along the axis **Z**, you can also click the **Operations >** Z↓ button in the **Properties** window of the **Plane**)

- Create a layer **Plot along curve** on the plane.
- In the **Properties** window of the **Plot along curve**, specify:

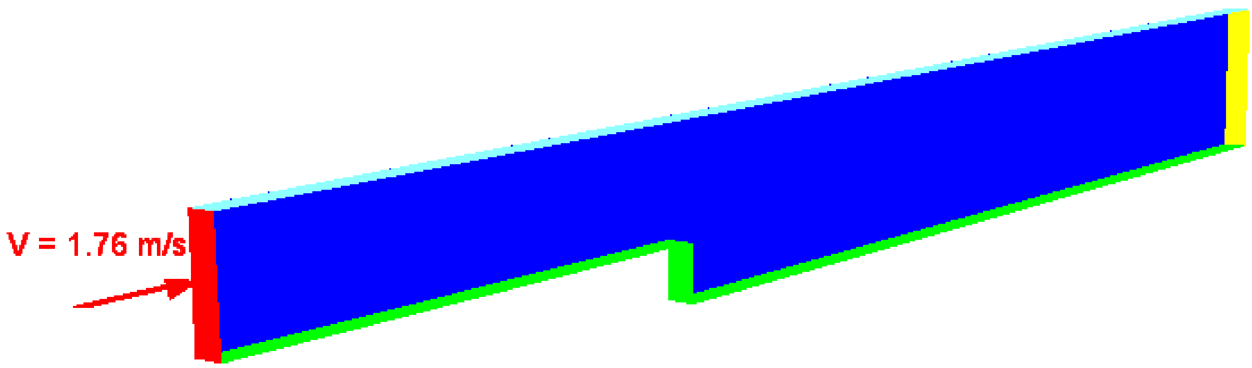
Variable

Variable	= **Shear stress**
Number of points	**= 1000**
Rotation angle	**= 180**

4.3.3 Turbulent flow around a backward facing step

The *Quadratic k-ε turbulence model* is intended for simulation of flows with values 30<y+<300 and recirculation zones.

Consider the use of the k-ε quadratic turbulence model in the example of a flow over an opposite facing step.



Dimensions:

Step height	D	= 1	[m]

Inflow parameters:

Velocity on inlet:	V_{inl}	= 1.76	[m s^{-1}]

Parameters of the substance:

Density	ρ	= 1	[kg m^{-3}]
Viscosity	μ	=1.82e-5	[kg m^{-1}s^{-1}]

Reynolds number: $$\mathrm{Re} = \frac{V_{inl} D \rho}{\mu} = \frac{1.76 \cdot 1 \cdot 1}{1.82 \cdot 10^{-5}} \approx 10^5$$

Geometry: `BackwardFacingStep.wrl`

Project: `BackwardFacingStep`

4.3.3.1 Physical model

In the folder **Substances**:

- Create **Substance #0**.
- Specify the following properties of the **Substance #0**:

Aggregative state	**= Gas**	
Molar mass		
Value	**= 0.0289**	[kg mole^{-1}]
Density		
Value	**= 1**	[kg m^{-3}]
Viscosity		
Value	**= 2e-5**	[kg m^{-1}s^{-1}]
Specific heat		
Value	**= 1009**	[J kg^{-1} K^{-1}]

In the folder **Phases**:

- Create a continuous **Phase #0**.
- In **Phase #0** add **Substance #0** into the folder **Substances**.
- Specify properties of the folder **Phase #0 > Physical processes**:

Motion	**= Navier-Stokes model**
Turbulence	**= KENL**

In the folder **Models**:

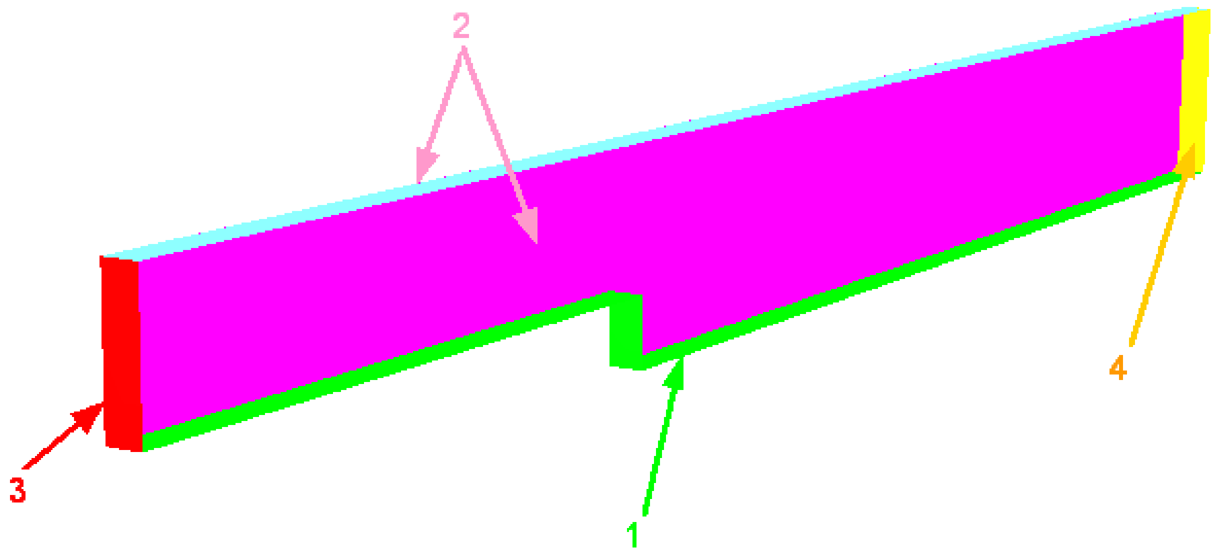
- Create **Model #0**.
- Add **Phase #0** into subfolder **Model #0 > Phases**
- In the folder **Init. data #0** specify:

Velocity (Phase #0)		
X	**= 1.76**	[m s^{-1}]
Y	**= 0**	[m s^{-1}]
Z	**= 0**	[m s^{-1}]
Pulsations (Phase #0)	**= 0.095**	
Turbulent scale (Phase #0)	**= 0.05**	[m]

4.3.3.2 Boundary conditions

In the **Properties** window of the **SubRegion #0**, specify:

Model = Model #0



Specify the following boundary conditions:

Boundary 1

Type	**= Wall**	
Variables		
Velocity (Phase #0)	**= Logarithm law**	
TurbEnergy (Phase #0)	**= Value in cell near wall**	
TurbDissipation (Phase #0)	**= Value in cell near wall**	

Boundary 2 (two lateral surfaces and the upper surface)

Type	**= Symmetry**	
Variables		
Velocity (Phase #0)	**= Slip**	
TurbEnergy (Phase #0)	**=Symmetry**	
TurbDissipation (Phase #0)	**=Symmetry**	

Boundary 3

Type	**= Inlet/Outlet**	
Variables		
Velocity (Phase #0)	**= Normal mass velocity**	
Mass velocity	**= 1.76**	[kg m^{-2} s^{-1}]
TurbEnergy (Phase #0)	**= Pulsations**	
Value	**= 0.095**	
TurbDissipation (Phase #0)	**= Turbulent scale**	
Value	**= 0.05**	[m]

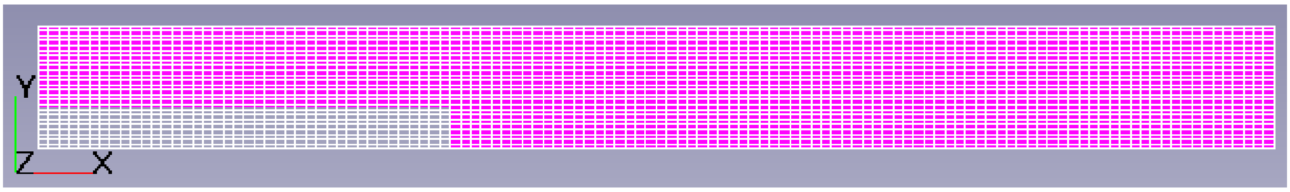
Boundary 4

Type	**= Free Outlet**	
Variables		
Velocity (Phase #0)	**= Pressure**	
Value	**= 0**	[Pa]

TurbEnergy (Phase #0) **= Zero gradient**

TurbDissipation (Phase #0) **= Zero gradient**

4.3.3.3 Initial grid



Specify in the **Properties** window of the **Initial grid**:

Grid structure = 2D

Plane = XY

nX = 120

nY = 25

In the **Properties** window of the **Initial grid** click **Apply**.

4.3.3.4 Parameters of calculation

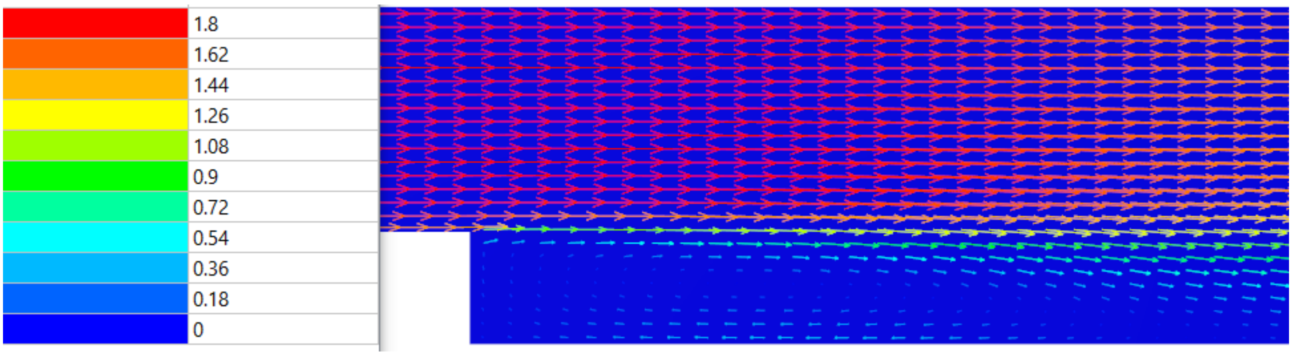
Specify in the **Solver** tab in properties of the **Time step** element:

Method	= **Via CFL number**	
Convective CFL	**= 100**	
Max step	**= 1**	[s]

4.3.3.5 Visualization

To view the dynamics of the solution during the computation, specify visualization of [**Velocity distribution**]() in the plane of the flow before the start of computation.

4.3.3.5.1 Velocity distribution



- In the **Properties** window of **Plane #0** specify:

Object

Normal

X **= 0**

Y **= 0**

Z **= 1**

(to direct a **Plane**'s normal along the axis **Z**, you can also click the **Operations >** Z↓ button in the **Properties** window of the **Plane**)

- Create a layer **Vectors** on **Plane #0**.

- In the **Properties** window of the layer specify:

On regular grid			**= No**
Coloring			
	Variable		
		Variable	**= Velocity**
	Value range		
		Mode	**= Manual**
		Max	**= 1.8**
		Min	**= 0**

The program will automatically specify the variable, which is used to build the vectors, **Variable > Variable = Velocity**.

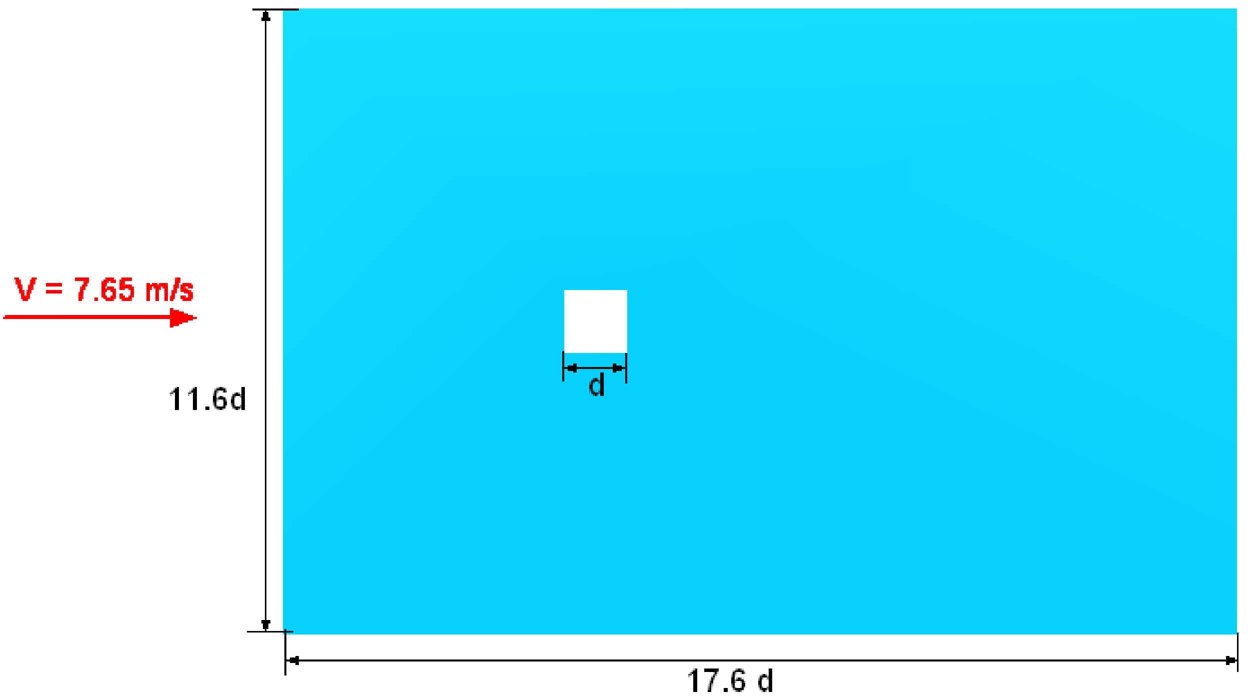
Note:

To display the palette, which is used for coloring the layer (in our case this is coloring the vectors by their absolute values), you can use the **Info** window or, in properties of the layer, set **Appearance > Enabled = Yes**. To open the **Info** window, you have to select the layer in the tree of **Postprocessor** and click the button.

4.3.4 Turbulent flow around a box

The turbulence model **SST** is used for simulation of flows with a high degree of turbulence, with recirculation zones and large back pressure gradients, and it also provides good results for free flows and flows with small pressure gradients.

Consider the use of the **SST** turbulence model in the example of a turbulent flow around a cube.



Dimensions:

Side of the cube	d	= 0.0254	[m]
Length of the computational domain	17.6d	= 0.447	[m]
Width of the computational domain	11.6d	= 0.294	[m]

Inflow parameters:

Velocity on inlet	V_{inl}	= 7.65	[m s^{-1}]
Parameters of the substance:			
Density	ρ	= 1.226	[kg m^{-3}]
Viscosity	μ	=1.8325e-5	[kg $m^{-1}s^{-1}$]

Reynolds number:

$$Re = \frac{V_{inl} d\rho}{\mu} = \frac{7.65 \cdot 0.0254 \cdot 1.226}{1.8325 \cdot 10^{-5}} \approx 1.3 \cdot 10^4$$

Geometry: **Box.STL**

Project: **Box**

4.3.4.1 Physical model

In the folder **Substances**:

- Create **Substance #0.**
- Specify the following properties of the **Substance #0**:

Aggregative state	**= Gas**	
Molar mass		
Value	**= 0.0289**	[kg $mole^{-1}$]
Density		
Value	**= 1.226**	[kg m^{-3}]
Viscosity		
Value	**= 1.82e-5** *)	[kg $m^{-1}s^{-1}$]
Specific heat		
Value	**= 1009**	[J kg^{-1} K^{-1}]

*) **1.82e-5** is notation for 1.82×10^{-5}.

In the folder **Phases**:

- Create a continuous **Phase #0.**
- In **Phase #0** add **Substance #0** into the folder **Substances.**
- Specify properties of the folder **Phase #0 > Physical processes**:

Motion	**= Navier-Stokes model**
Turbulence	**= SST**

In the folder **Models**:

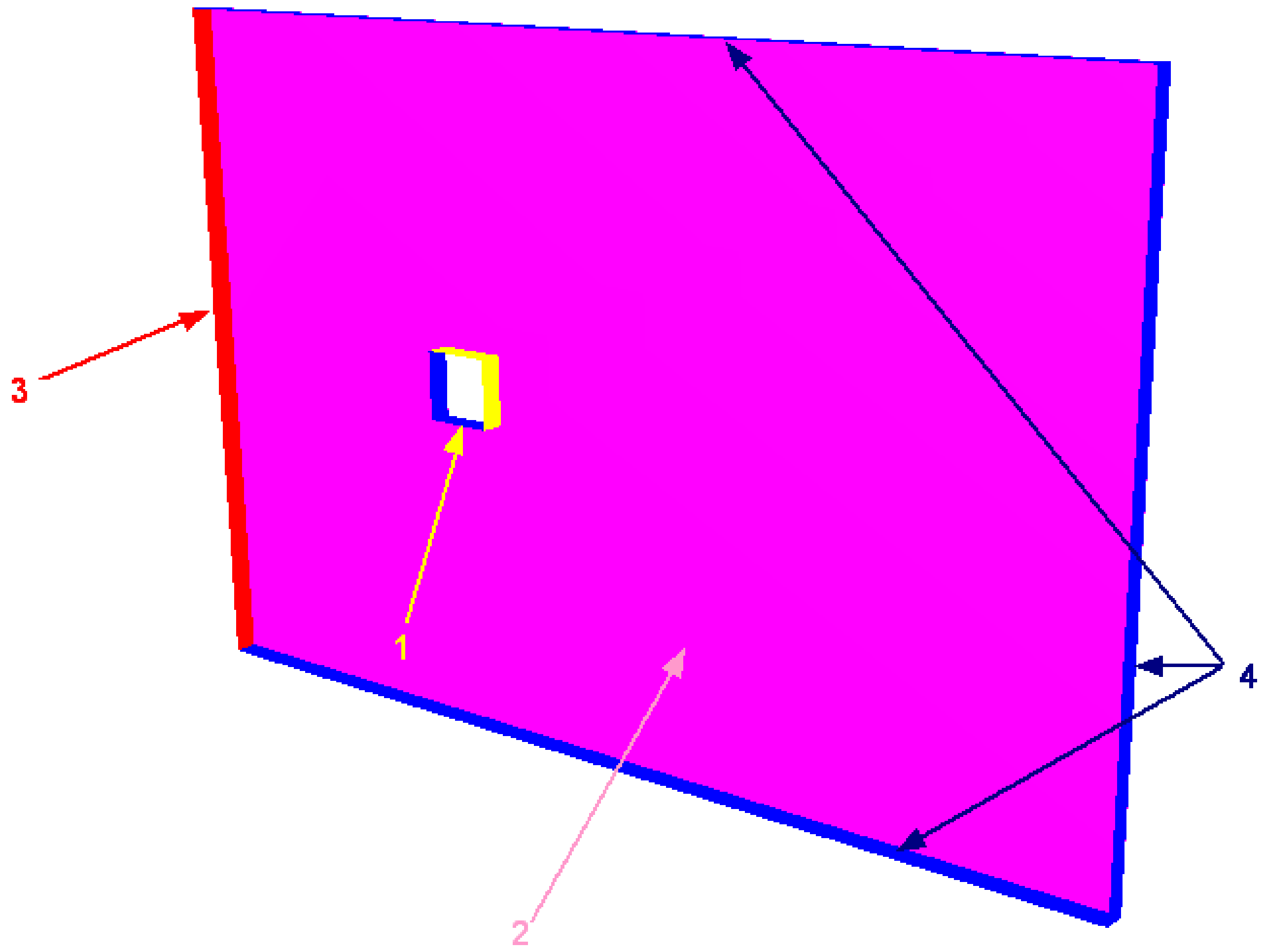
- Create **Model #0**.
- Add **Phase #0** into subfolder **Model #0 > Phases**
- In the folder **Init. data #0** specify:

Velocity (Phase #0)		
X	**= 7.65**	[m s^{-1}]
Y	**= 0**	[m s^{-1}]
Z	**= 0**	[m s^{-1}]
Pulsations (Phase #0)	**= 0.05**	
Turbulent scale (Phase #0)	**= 0.00254**	[m]

4.3.4.2 Boundary conditions

In the **Properties** window of the **SubRegion #0**, specify:

Model = Model #0



Specify the following boundary conditions:

Boundary 1

Type	**= Wall**
Wall interaction	
Phase #0	**=No wall functions**
Variables	
Velocity (Phase #0)	**= No slip**
TurbEnergy (Phase #0)	**= Fixed value**
TurbDissipation (Phase #0)	**= Value in cell near wall**

Boundary 2

Type	**= Symmetry**
Variables	
Velocity (Phase #0)	**= Slip**
TurbEnergy (Phase #0)	**=Symmetry**
TurbDissipation (Phase #0)	**=Symmetry**

Boundary 3

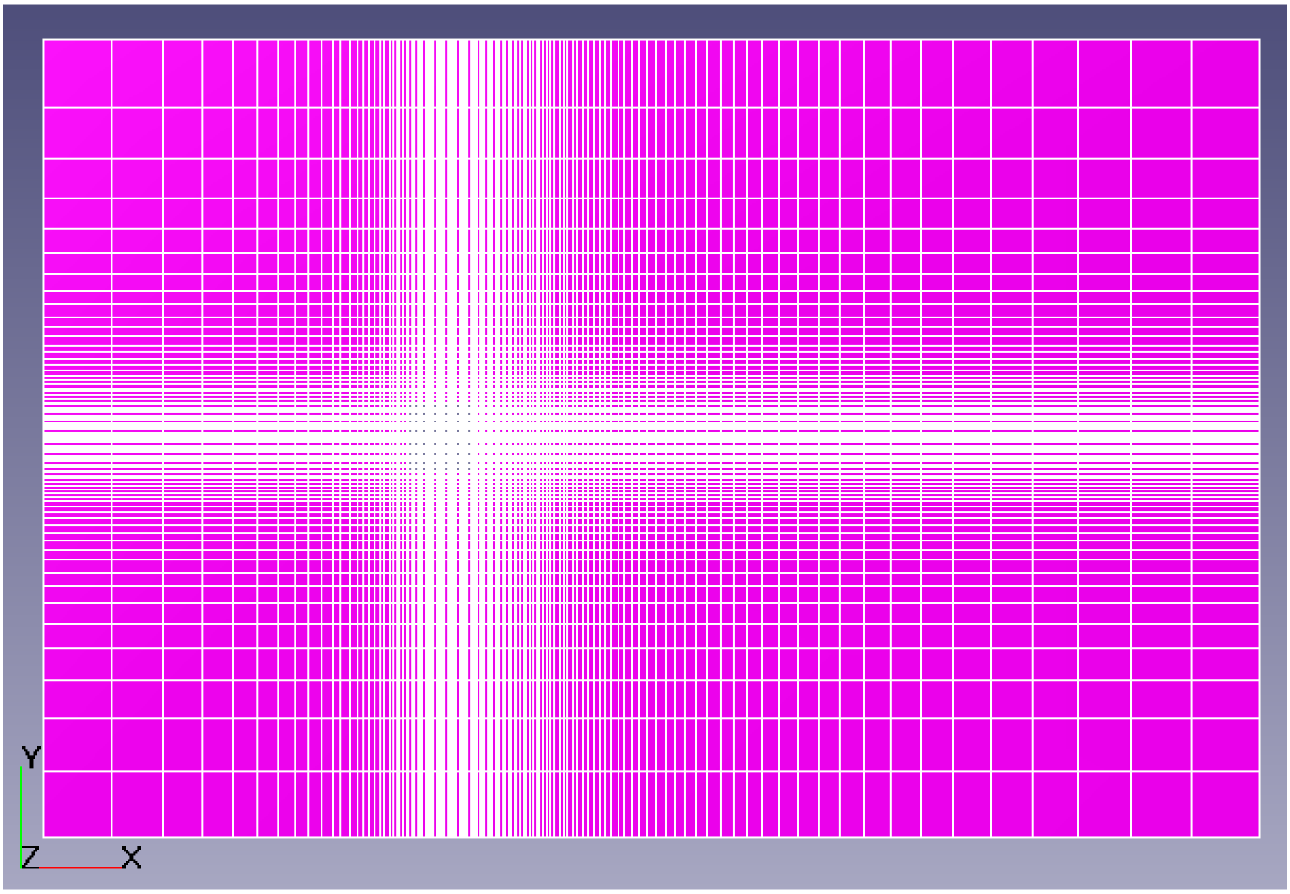
Type	**= Inlet/Outlet**	
Variables		
Velocity (Phase #0)	**= Normal mass velocity**	
Mass velocity	**= 9.38**	[kg m^{-2} s^{-1}]
TurbEnergy (Phase #0)	**= Pulsations**	
Value	**= 0.05**	
TurbDissipation (Phase #0)	**= Turbulent scale**	
Value	**= 0.00254**	[m]

(Here values of values of parameters of turbulence of the approach flow are set because it is planned to do computation using the SST turbulence model without wall functions)

Boundary 4

Type	**= Free Outlet**	
Variables		
Velocity	**= Pressure**	
Value	**= 0**	[Pa]
TurbEnergy	**= Zero gradient**	
TurbDissipation	**= Zero gradient**	

4.3.4.3 Initial grid



In the **Properties** window of the **Initial grid**, click the button to open the **Initial grid editor.**

Specify in the **Initial grid editor**:

for axis OX

Grid parameters

h_max = **0.03** [m]

h_min = **0.00075** [m]

Insert a reference line with coordinate **x=0** [m].

Specify **Reference line parameters** for the reference line with coordinate **x=-0.147**:

h = **0.03** [m]

Specify **Reference line parameters** for the reference line with coordinate **x=0**:

h **=0.00075** [m]

kh- **= 1**

kh+ **= 1**

Specify **Reference line parameters** for the reference line with coordinate **x=0.3**:

h = **0.03** [m]

for axis OY (click the button )

Grid parameters

h_max = **0.03** [m]

h_min = **0.00075** [m]

Insert a reference line with coordinate **y=0** [m].

Specify **Reference line parameters** for the reference line with coordinate **y=-0.147**:

h = **0.03** [m]

Specify **Reference line parameters** for the reference line with coordinate **y=0**:

h **=0.00075** [m]

kh- **= 1** [m]

kh+ **= 1** [m]

Specify **Reference line parameters** for the reference line with coordinate **y=0.147**:

h = **0.03** [m]

Click **OK** to close the **Initial grid editor** with saving the entered data.

In properties of the **Initial grid** specify:

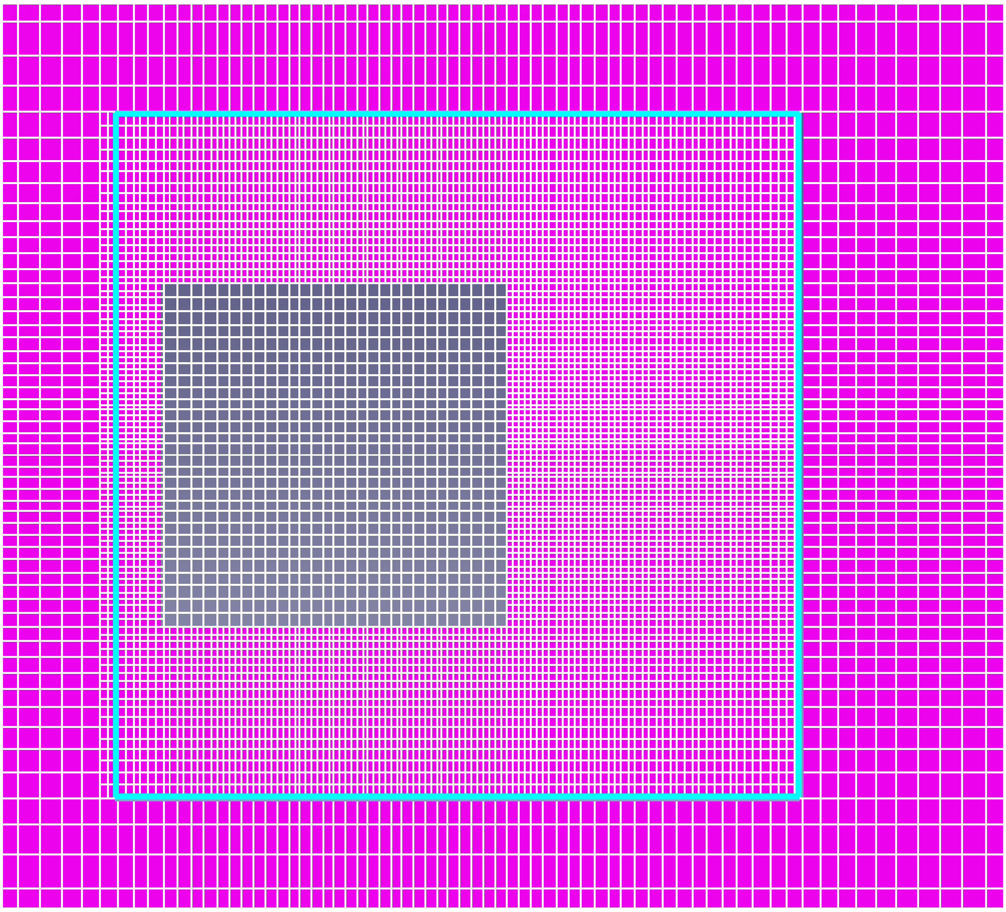
Grid structure = 2D

Plane = XY

In the **Properties** window of the **Initial grid** click **Apply**.

4.3.4.4 Adaptation of the computational grid

In this example, you must resolve the vortex formation zone. For this it is necessary to make a grid adaptation in the volume of a parallelepiped around the cube (you can see cells of the adapted computational grid after the computation using the **Computational grid** layer).



Specifying the grid adaptation in an object consists of two steps:

1. Specifying an **Object** for the adaptation
2. Specifying adaptation criteria

In order to create an **Object** for the adaptation:

- From the context menu of the folder **Objects**, select **Create**.
- In the **Create new object** window select **Object type** = **Box**.
- In the **Properties** window of the **Box #0** specify:

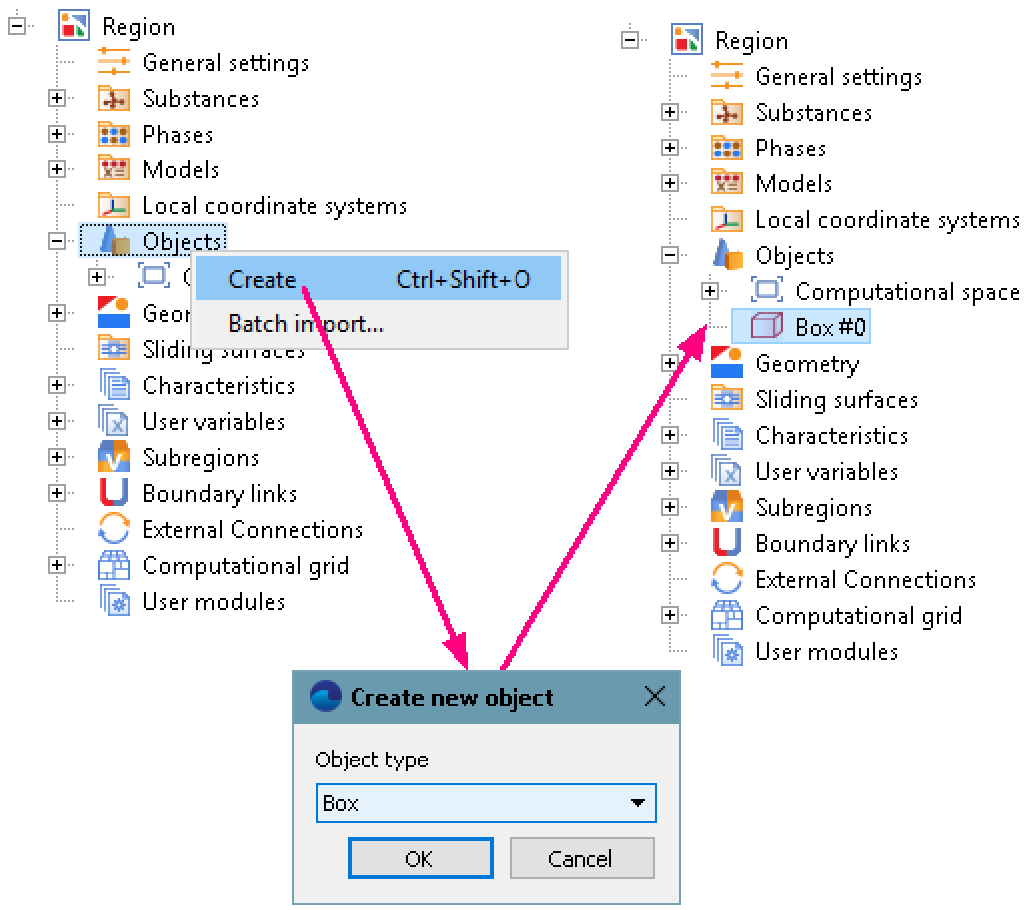
Location

Reference point

X	**= 0.01**	[m]
Y	**= 0**	[m]
Z	**= 0.005**	[m]

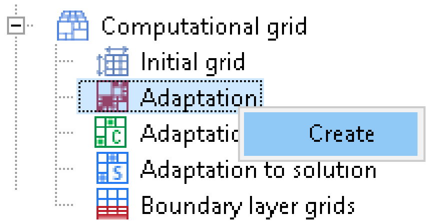
Size

X	**= 0.05**	[m]
Y	**= 0.05**	[m]
Z	**= 0.005**	[m]



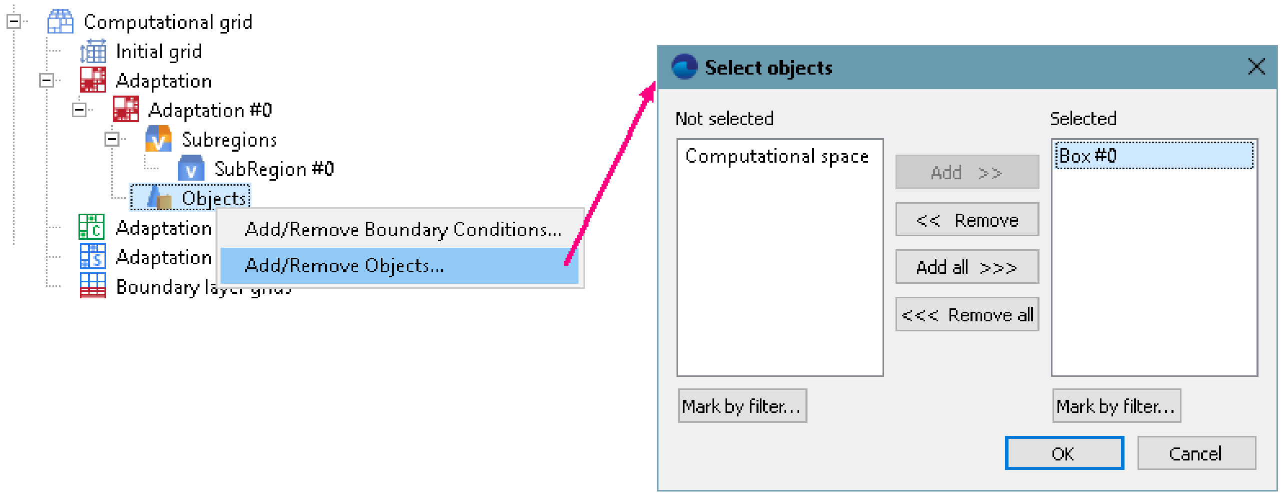
Set up the adaptation:

- From the context menu of the folder **Computational grid > Adaptation**, select the **Create** command to create a new element **Adaptation #0**:



> As in this exercise there is only one **Subregion**, you don't have to add **SubRegion #0** into the folder **Computational grid > Adaptation > Adaptation #0 > Subregions** using the **Add/Remove** command from the context menu; **SubRegion #0** is added to this folder automatically at creation of **Adaptation #0**.

- From the context menu of the element **Adaptation #0 > Objects** select the command **Add/Remove Objects** and in the **Select objects** dialog box, which opens, place **Box #0** into the pane **Selected** and click **OK**:



- In the **Properties** window of the new created element **Adaptation #0** specify:

Enabled	**= Yes**
Max level N	**= 1**
Split/Merge	**= Split**
Layers	
Layers for Level N	**= 4**

Note:

To view the computational grid adaptation, create a layer **Computational grid section** on **Plane #0**.

4.3.4.5 Parameters of calculation

Specify in the **Solver** tab in properties of the **Time step** element:

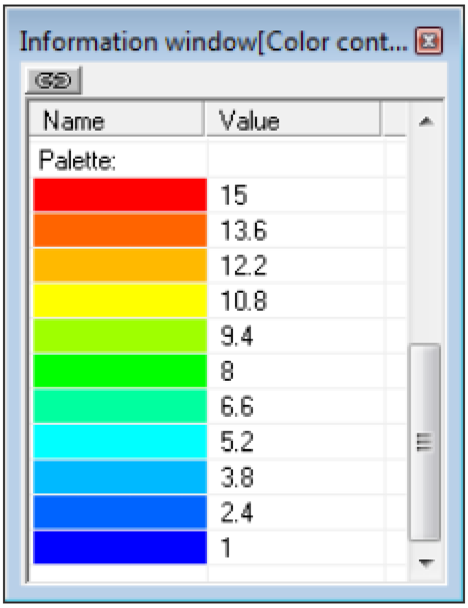
Method	**= In seconds**	
Constant step	**= 0.0003**	[s]

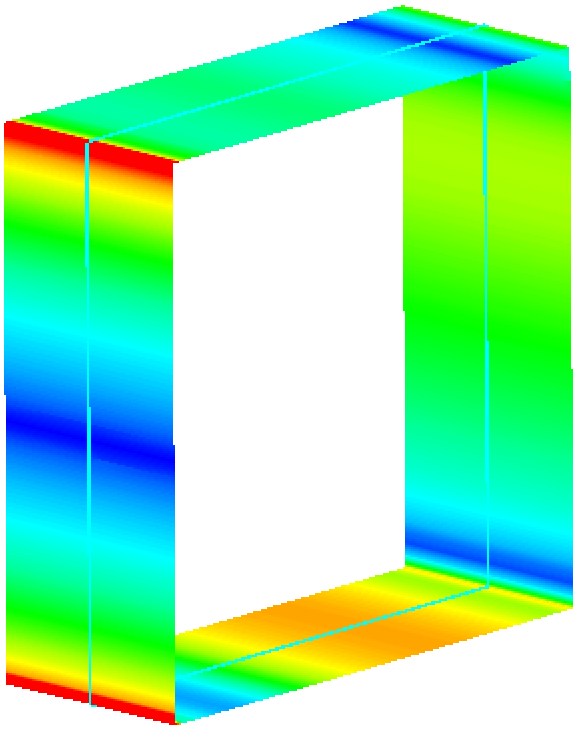
4.3.4.6 Visualization

To view the dynamics of the solution during the computation, prior to computation, specify visualization of:

1. **Y+ distribution** on the surface of the parallelepiped.
2. **Velocity distribution** in the plane of symmetry.

4.3.4.6.1 Y+ distribution





- Create a **Supergroup** on the BC **Wall** using the **Create supergroup > In Postprocessor** command from the context menu.
- Create a layer **Color contours** on the **Supergroup**.
- In the **Properties** window of the layer specify:

Variable

Variable	**= Y_plus**

Value range

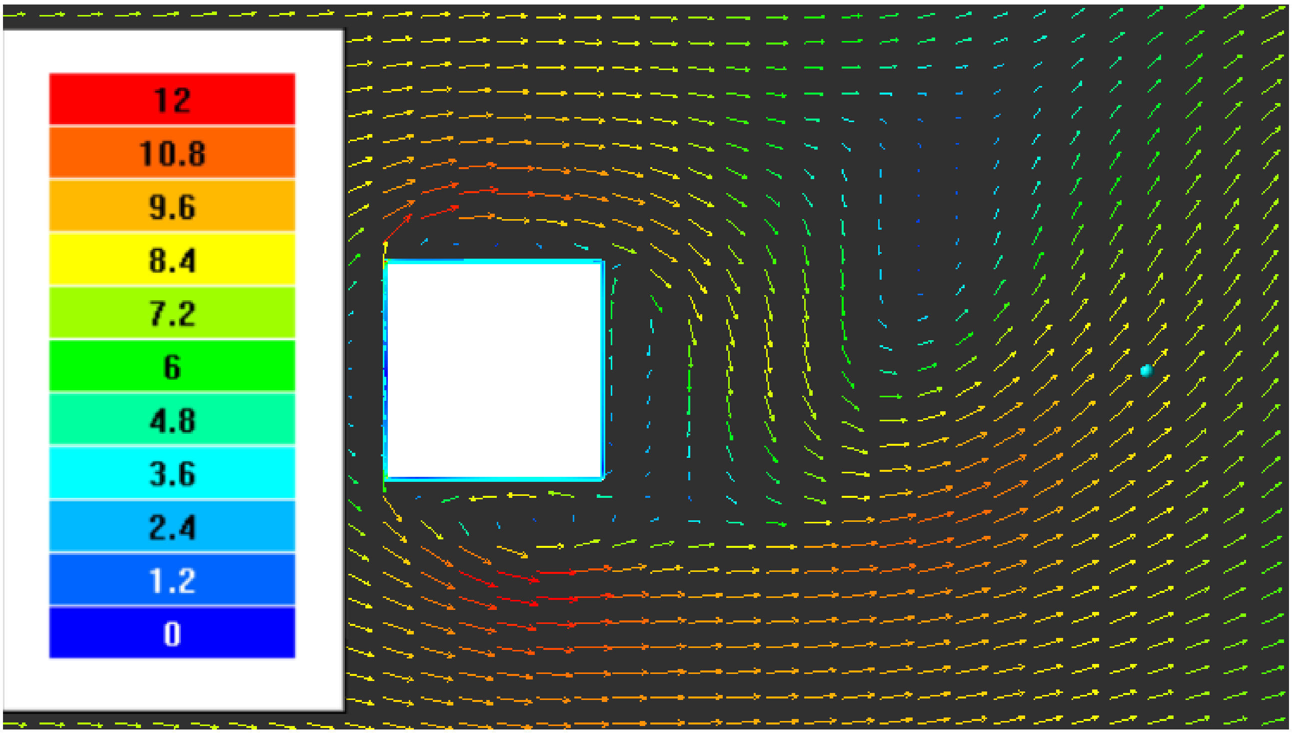
Mode	**= Manual**
Max	**= 15**
Min	**= 1**

Notes:

- To display the palette, which is used for coloring the layer, you can use the **Info** window or, in properties of the layer, set **Appearance > Enabled = Yes**. To open the **Info** window, you have to select the layer in the tree of **Postprocessor** and click the button.
- If you wish to hide some layers or objects displayed in the **View** window, use the **Hide** command from their context menus.
- Computation of this project may require long time.

4.3.4.6.2 Velocity distribution

Visualization of velocity distribution when the step number is near 1100:



Velocity distribution visualized as vectors displays generation of turbulent eddies, which form and go away from the box some time after starting the computation (initially the fluid moves behind the box symmetrically and with no eddies)

- In the **Properties** window of **Plane #0** specify:

Object

Normal

X	**= 0**
Y	**= 0**
Z	**= 1**

(to direct a **Plane**'s normal along the axis **Z**, you can also click the **Operations >** button in the **Properties** window of the **Plane**)

- Create a layer **Vectors** on **Plane #0**.
- In the **Properties** window of the layer specify:

Grid

Size 1	**= 100**
Size 2	**= 100**

Coloring

Variable

Variable	**= Velocity**

Value range

Mode	**= Manual**
Max	**= 12**
Min	**= 0**

The program will automatically specify the variable, which is used to build the vectors, **Variable > Variable = Velocity**.

Notes:

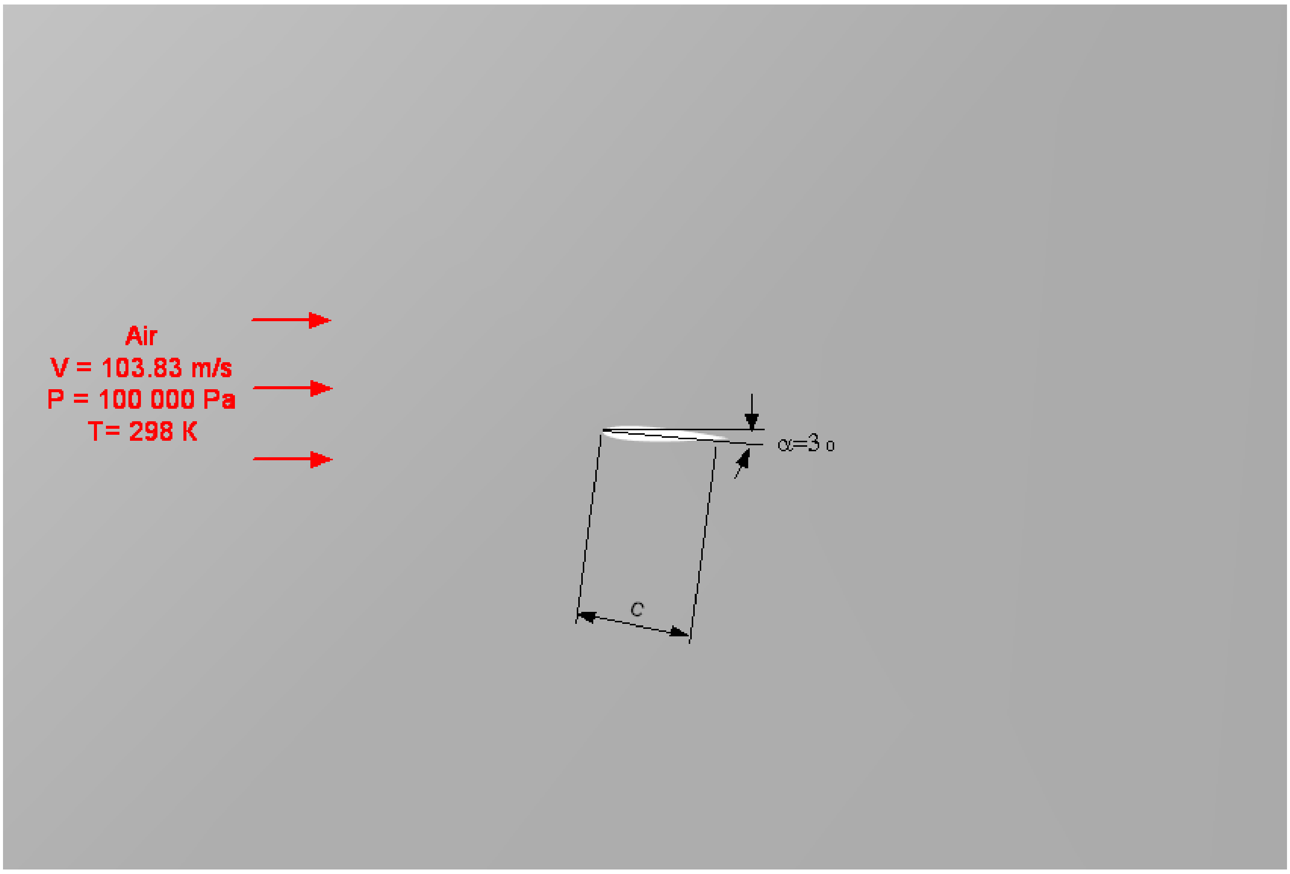
- To display the palette, which is used for coloring the layer (in our case this is coloring the vectors by their absolute values), you can use the **Info** window or, in properties of the layer, set **Appearance > Enabled = Yes**. To open the **Info** window, you have to select the layer in the tree of **Postprocessor** and click the button.
- If you wish to hide some layers or objects displayed in the **View** window, use the **Hide** command from their context menus.
- Computation of this project may require long time.

4.3.5 Subsonic flow around an airfoil

The SA turbulence model is an one-parameter model, which was developed for aerospace applications. This model can be used both in Low-Reynolds and Hi-Reynolds computations.

Low-Reynolds computations assume that the viscous boundary layer is resolved by a computational grid. We recommend to build a grid with dimensionless distance y+ from the center of the wall cell to the wall does not exceed 1 (the structure of the boundary layer is discussed in work H.Schlichting (1974) "Boundary layer theory" Nauka, Moscow, 711 pages / Шлихтинг Г. (1974) "Теория пограничного слоя", Москва, Наука, 711 с.).

Consider the application of the SA model for the example problem of subsonic flow around airfoil at Reynolds number Re = 1.68×10^6 and Mach number M = 0.3. In this example Hi-Reynolds computations are done, so a quite coarse computational grid is built (the viscous boundary layer is not resolved) and wall functions are used.



Dimensions:

Chord length	c	= 0.256	[m]
Dimensions of the computational domain		5.3 × 5 × 0.00254	[m × m × m]
Angle of attack		α= 3 °	

Substance: Air

Inflow parameters:

Static pressure:	P	= 100000	[Pa]

Static temperature:	T	= 298	[K]
Velocity on inlet:	V_{inl}	= 103.83	[m s^{-1}]
Mach number:	M	= 0.3	
Reynolds number:	Re	= 1.68×10^6	

Geometry: `NACA0012_3deg.STL`

Project: `NACA0012_3deg`

Note:

Computation of this project may require significant computing resources and long time.

4.3.5.1 Physical model

In the **Properties** window of the element **General settings** specify:

Reference values

Temperature	**= 298**	[K]
Pressure	**= 100000**	[Pa]

In the folder **Substances**:

- Create **Substance #0**.
- Load **Substance #0** from the **Standard** substance database:
  - From the context menu of **Substance #0** select **Load from SD > Standard**.
  - In the new window **Load from database**, select:

Substances	**= Air**
Phases	**= Gas (equilibrium)**

In the folder **Phases**:

- Create a continuous **Phase #0**.
- Add the **Air_Gas (equilibrium)** substance into the folder **Phase #0 > Substances**.
- Specify properties of the folder **Physical processes**:

Heat transfer	**= Heat transfer via H**
Motion	**= Navier-Stokes model**
Turbulence	**= SA**

In the folder **Models**:

- Create **Model #0**.
- Add **Phase #0** into subfolder **Model #0 > Phases**
- In the folder **Init. data #0** specify:

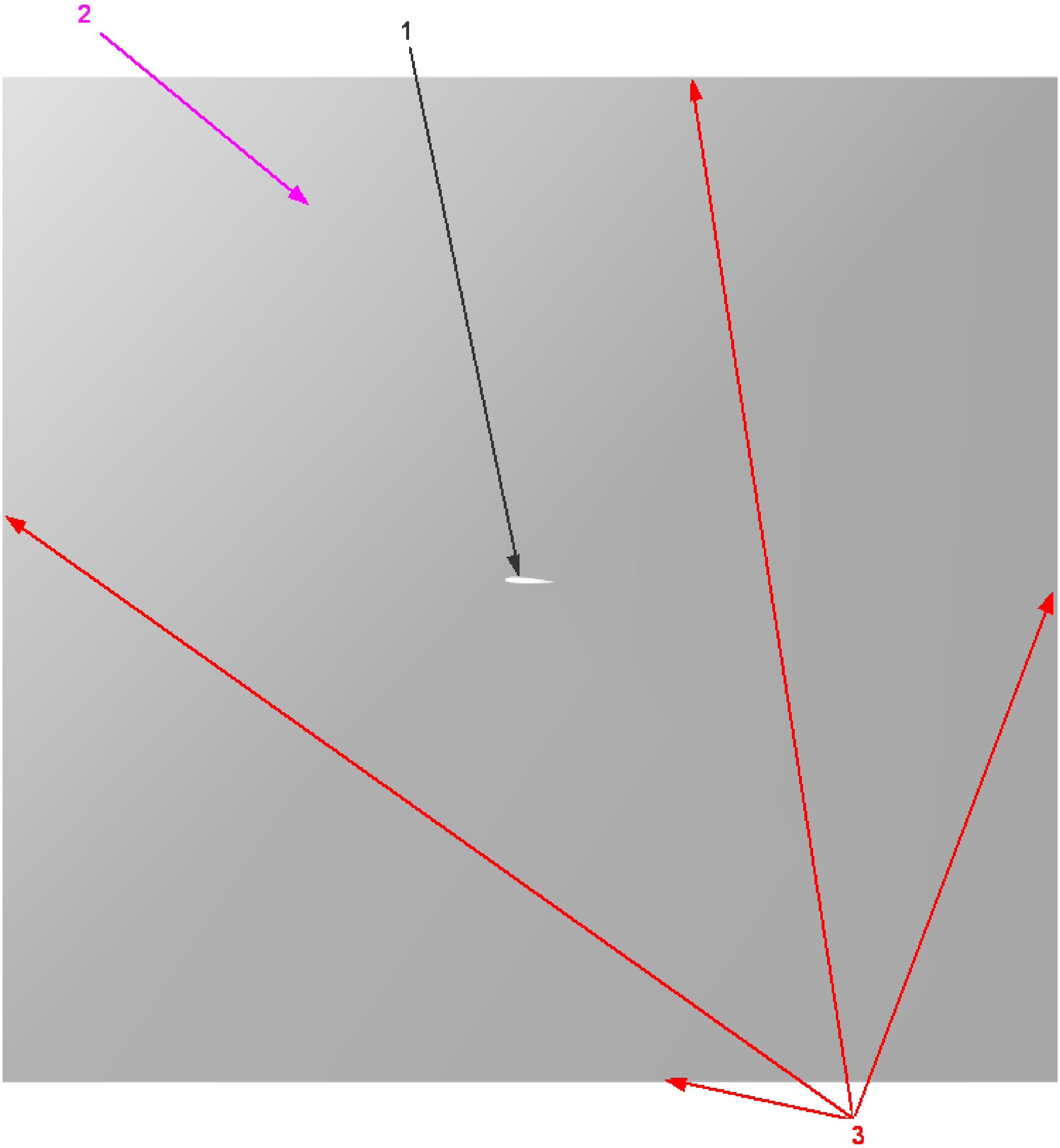
Velocity

X	**= 103.83**	[m s^{-1}]
Y	**= 0**	[m s^{-1}]
Z	**= 0**	[m s^{-1}]

4.3.5.2 Boundary conditions

In the **Properties** window of the **SubRegion #0**, specify:

Model = Model #0



Boundary 1

Type	= **Wall**
Variables	
Temperature(Phase #0)	= **Zero gradient**
Velocity(Phase #0)	= **Logarithm law**
TurbKinViscosity(Phase #0)	= **Value in cell near wall**

Boundary 2

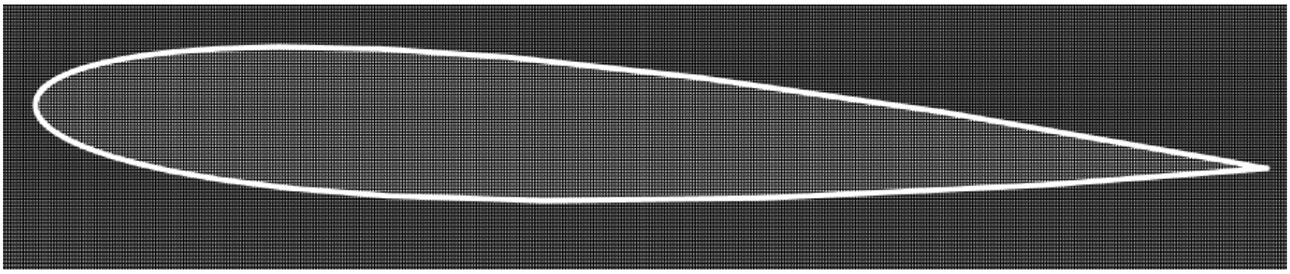
Type	= **Symmetry**
Variables	
Temperature(Phase #0)	= **Symmetry**
Velocity(Phase #0)	= **Slip**

TurbKinViscosity(Phase #0)	**= Symmetry**	

Boundary 3

Type	= **Non-reflecting**	
Variables		
Temperature(Phase #0)	= **Non-reflect.**	
Value	**= 0**	[K]
Velocity(Phase #0)	= **Non-reflect.**	
Velocity at inf.		
X	**= 103.83**	[m s^{-1}]
Y	**= 0**	[m s^{-1}]
Z	**= 0**	[m s^{-1}]
Pressure at inf.	**= 0**	[Pa]
TurbKinViscosity(Phase #0)	**= Value**	
Value	**= 0**	[m^2 s^{-1}]

4.3.5.3 Initial grid



In the **Properties** window of the **Initial grid**, click the button to open the **Initial grid editor**.

Specify in the **Initial grid editor**:

for axis OX

Grid parameters

h_max **= 0.634** [m]

h_min **= 0.0006415** [m]

Insert a reference line with coordinate **x=0** [m].

Specify **Reference line parameters** for the reference line with coordinate **x=-2.54**: [m]

h **= 0.634** [m]

Specify **Reference line parameters** for the reference line with coordinate **x=0**: [m]

h **=0.0006415** [m]

kh- **= 1**

kh+ **= 0.95**

Specify **Reference line parameters** for the reference line with coordinate **x=2.794**: [m]

h **= 0.634** [m]

for axis OY (click the button Y)

Grid parameters

h_max **= 0.634** [m]

h_min **= 0.0006415** [m]

Insert a reference line with coordinate **y=0** [m].

Specify **Reference line parameters** for the reference line with coordinate **y=-2.54**:

h	**=0.634**	[m]

Specify **Reference line parameters** for the reference line with coordinate **y=0**: [m]

h	**=0.0006415**	[m]
kh-	**= 1**	
kh+	**= 1**	

Specify **Reference line parameters** for the reference line with coordinate **y=2.54**: [m]

h	**=0.634**	[m]

Click **OK** to close the **Initial grid editor** with saving the entered data.

In properties of the **Initial grid** specify:

Grid structure = 2D

Plane = XY

In the **Properties** window of the **Initial grid** click **Apply**.

4.3.5.4 Adaptation of the computational grid

Specify adaptation on the surface of the wing:

- From the context menu of the folder **Adaptation**, select the **Create** command to create **Adaptation #0**.
- From the context menu of the element **Adaptation #0 > Objects** select the command **Add/Remove Boundary Conditions** and in the **Select boundary conditions** dialog box, which opens, place the **Boundary condition**, which correspond to the wall, into the pane **Selected** and click **OK**.
- In the **Properties** window of the element **Adaptation #0** specify:

Enabled	**= Yes**
Max level N	**= 3**
Layers	
Layers for Level N	**= 5**
Layers for Level N-1	**= 4**
Layers for Level N-2	**= 4**

To make parameters **Layers for Level N-1** and **Layers for Level N-2** available, select in the **Properties** window the **Layers** line and twice click there the screen button  **(Append item to the array)**.

4.3.5.5 Parameters of calculation

Specify in the **Solver** tab in properties of the **Time step** element:

Method	**=In seconds**	
Constant step	**= 1e-5** *)	[s]

*) This notation means 10^{-5}.

Note:

In this problem, the time step is chosen approximately equal to 0.04 of the time of flight over the wing's chord.

$$\tau_{own} = 0.04 * \frac{L}{V} = 0.04 * \frac{0.256}{103.83} \approx 10^{-5}$$

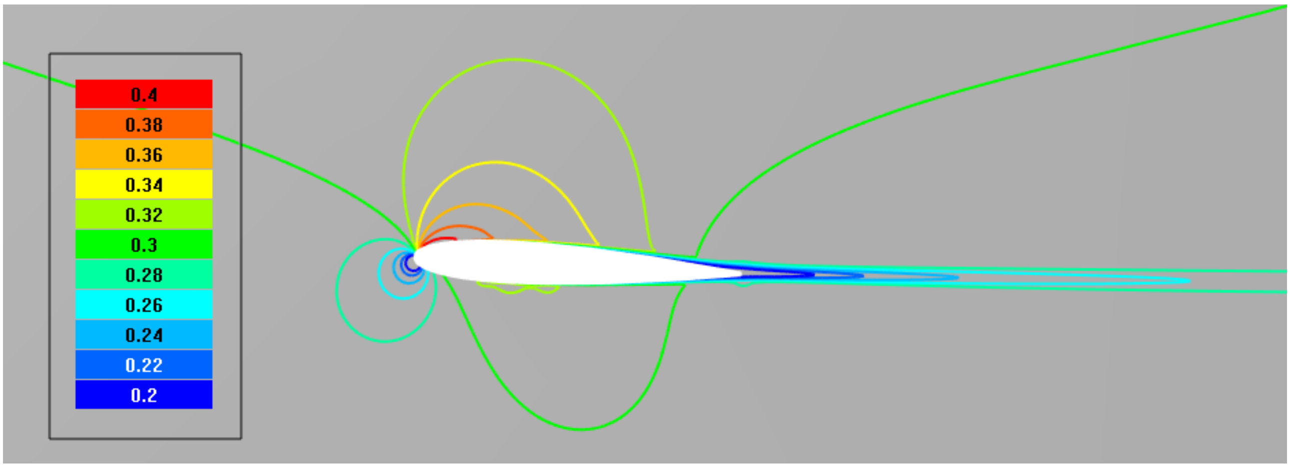
4.3.5.6 Visualization

To view the dynamics of the solution during the computation, prior to computation, specify visualization of:

1. **Mach Number distribution** in the plane of the flow.
2. **Pressure distribution** on the surface of the wing.

4.3.5.6.1 Mach Number distribution

Visualization at the step number 5000:



- In the **Properties** window of **Plane #0** specify:

Object
  Normal
    X **= 0**
    Y **= 0**
    Z **= 1**

(to direct a **Plane**'s normal along the axis **Z**, you can also click the **Operations >** button in the **Properties** window of the **Plane**)

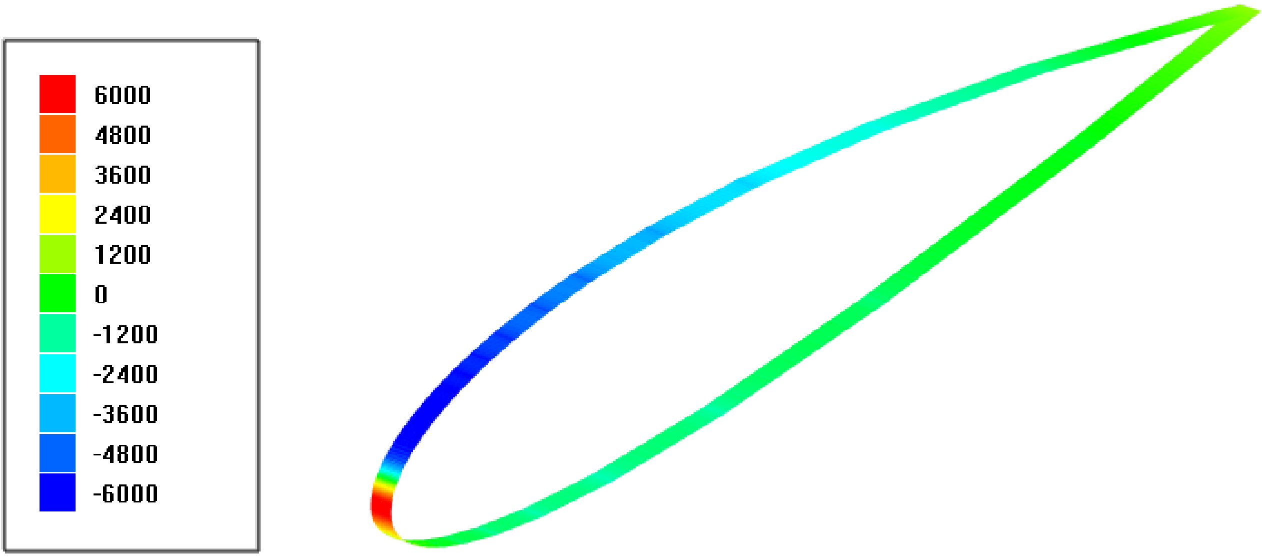
- Create a layer **Color contours** on **Plane #0**.
- In the **Properties** window of the **Color contours** specify:

Variable
  Variable **= MachNumber**
Value range
  Mode **= Manual**
  Max **= 0.4**
  Min **= 0.2**
Method = **Isolines**
Palette
  Appearance
    Enabled **= Yes**
    Style **= Style 1**

Notes:

- To display the palette, which is used for coloring the layer, you can use the **Info** window or, in properties of the layer, set **Appearance > Enabled = Yes**. To open the **Info** window, you have to select the layer in the tree of **Postprocessor** and click the button.
- If you wish to hide some layers or objects displayed in the **View** window, use the **Hide** command from their context menus.

4.3.5.6.2 Pressure distribution



- Create a **Supergroup** on the BC **Wall** using the command **Create supergroup > In Postprocessor** from the context menu.
- Create a layer **Color contours** on the **Supergroup**.
- In the **Properties** window of the **Color contours** specify:

Variable		
	Variable	**= Pressure**
Shift	**= 1e-5**	
Value range		
	Mode	**= Manual**
	Max	**= 6000**
	Min	**= -6000**

Notes:

- To display the palette, which is used for coloring the layer, you can use the **Info** window or, in properties of the layer, set **Appearance > Enabled = Yes**. To open the **Info** window, you have to select the layer in the tree of **Postprocessor** and click the button.
- If you wish to hide some layers or objects displayed in the **View** window, use the **Hide** command from their context menus.

4.4 Mass transfer

In *FlowVision* the following mass transfer models are implemented:

- **Mixing** simulates convective-diffusive mixing of several **Substances** including simple chemical reactions and radioactive decay.
- **Chemistry** simulates convective-diffusive mixing of several **Substances** with chemical reaction(s)
- **Combustion** simulates gas-phase combustion

Simulation of the mass transfer generally also assumes simulation of physical processes **Motion** and **Heat transfer**, which are turned on by similarly-named parameters of the element **Phase #N > Physical processes** in the project tree.

To simulate the mass transfer you have to start with:

- Specify **Substances**, which will be used in the mass transfer, and you have to specify their **Aggregative states** as either **Liquids** or **Gases**.
- Add no less then two of these **Substances** into the simulated **Phase #N**. When **Mixing** is simulated, it is recommended to place the **Substance**, which has higher mass fraction in the **Phase**, on the last place in the folder **Phase #N > Substances**.
- In properties of each of these **Substances** you have to specify values of their **Molar mass** and **Density**.
- Specify the model of mass transfer (select from a drop-down list the value of the **Mass transfer** parameter in properties of the element **Phase #N > Physical processes** in the project tree).
- Specify parameters of the mass transfer that don't depend on the mass transfer's model (**Time step coefficient**, **Explicit scheme**, ρD or array of the Schmidt numbers), specify use of an **Ablation** model (simulation of ablation requires enabling the physical process **Heat transfer**).
- Specify parameters **D** and **F** in the **Properties** windows of the elements **Mass transfer > Substances > *Substance*** (if **Mixing** or **Combustion** is simulated).

When **Chemistry** or **Combustion** is simulated, you have also to specify other parameters of mass transfer.

When **Chemistry** is simulated, you have to specify:

- the **Substances** that take part in the chemical reactions (**Phase #N > Physical processes > Mass transfer > Elements > *Substances***) and their parameters
- chemical reactions (as elements **Phase #N > Physical processes > Mass transfer > Reactions > Reaction #N**) and their parameters (rates of the forward and reverse reactions, stoichiometric coefficients, coefficients of efficiencies of the **Substances** in the dissociation-recombination reactions)
- initial and boundary conditions on concentrations of the **Substances**

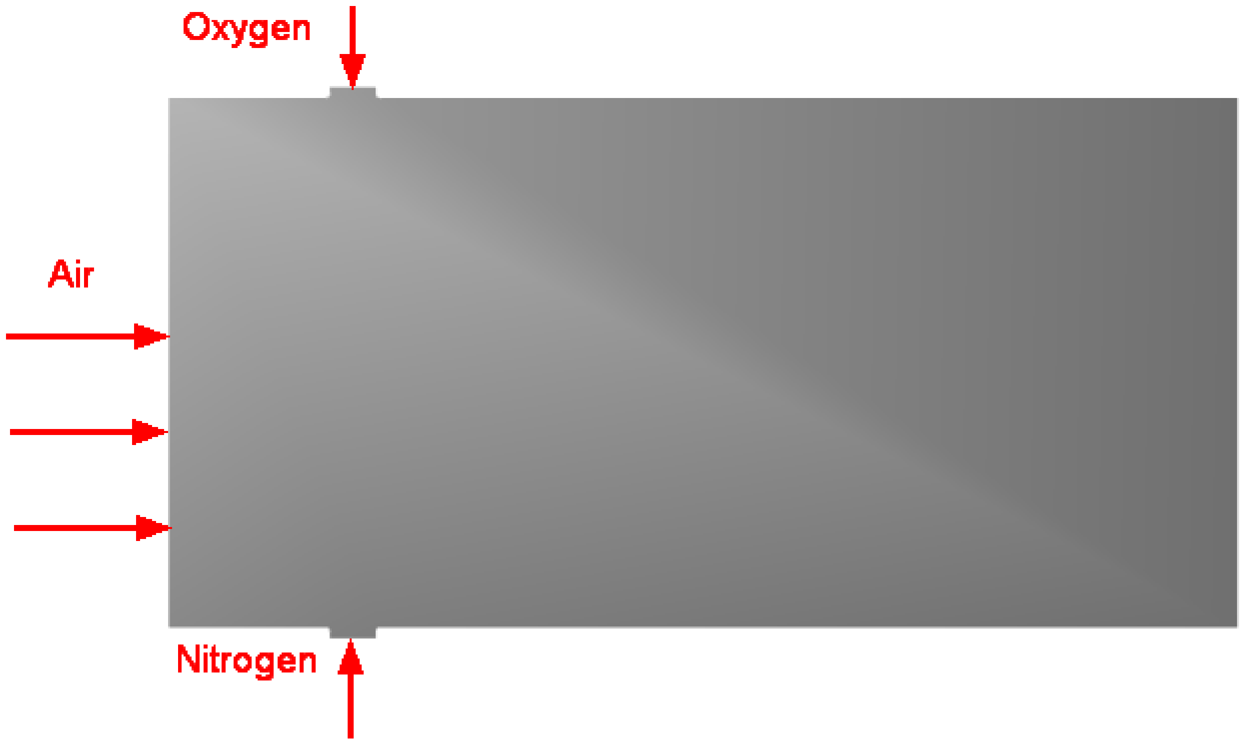
When **Combustion** is simulated, you have to specify:

- which **Substances** are fuel, oxidizer and products of the combustion
- the model of combustion and its parameters (stoichiometric coefficients **i_1**, **i_2**, **i_3**, temperature of ignition **T ignition**, lower and upper combustion limits **Alpha min.** and **Alpha max.**)
- initial and boundary conditions on concentrations of the fuel, oxidizer and products of the combustion, and variance of fuel (if **Arrhenius-Magnussen** combustion model is used).
- modifier(s) **Ignition / extinction zone** if you need to specify initiation of the combustion or specify zone(s) of extinction

To prevent a possible instability of the solution, we recommend to specify constraints for the computation.

4.4.1 Mixing of non-reacting substances

Consider simulation of mixing air with pure oxygen and pure nitrogen.



Parameters of the problem setting

Inflow parameters:

Mass flow rate of air	V_{air}	= 0.129	[kg $m^{-2}s^{-1}$]
Mass flow rate of oxygen	V_{O2}	= 1.4	[kg $m^{-2}s^{-1}$]
Mass flow rate of nitrogen	V_{N2}	= 1.24	[kg $m^{-2}s^{-1}$]

Geometry file and project's name

Geometry: `Mixture.wrl`

Project: `Mixture`

4.4.1.1 Physical model

In the folder **Substances**:

- Create **Substance #0**.
- Load **Substance #0** from the **Standard** substance database:
  - From the context menu of **Substance #0** select **Load from SD > Standard**.
  - In the new window **Load from database**, select:

    Substances = **Air**

    Phases = **Gas (equilibrium)**
- Create **Substance #0**.
- Load the properties of **Substance #0** from the **Substance Database**:

  Substances = **Oxygen**

  Phases = **Gas (equilibrium)**

- Create **Substance #0.**
- Load the properties of **Substance #0** from the **Substance Database**:

Substances	**= Nitrogen**
Phases	**= Gas**

In the folder **Phases**:

- Create a continuous **Phase #0.**
- In **Phase #0** add all **Substances**, which you have just created, into the folder **Substances** in the following order:
    - **Oxygen_Gas (equilibrium)**
    - **Nitrogen_Gas**
    - **Air_Gas (equilibrium)**

  When **Mixing** is simulated, we recommend to place the **Substance**, which has higher mass fraction in the **Phase**, on the last place in the folder **Phase #N > Substances**.
- In the **Properties** window of the folder **Phase #0 > Physical processes** specify:

Motion	**= Navier-Stokes model**
Mass transfer	**= Mixing**
Turbulence	**= KES**

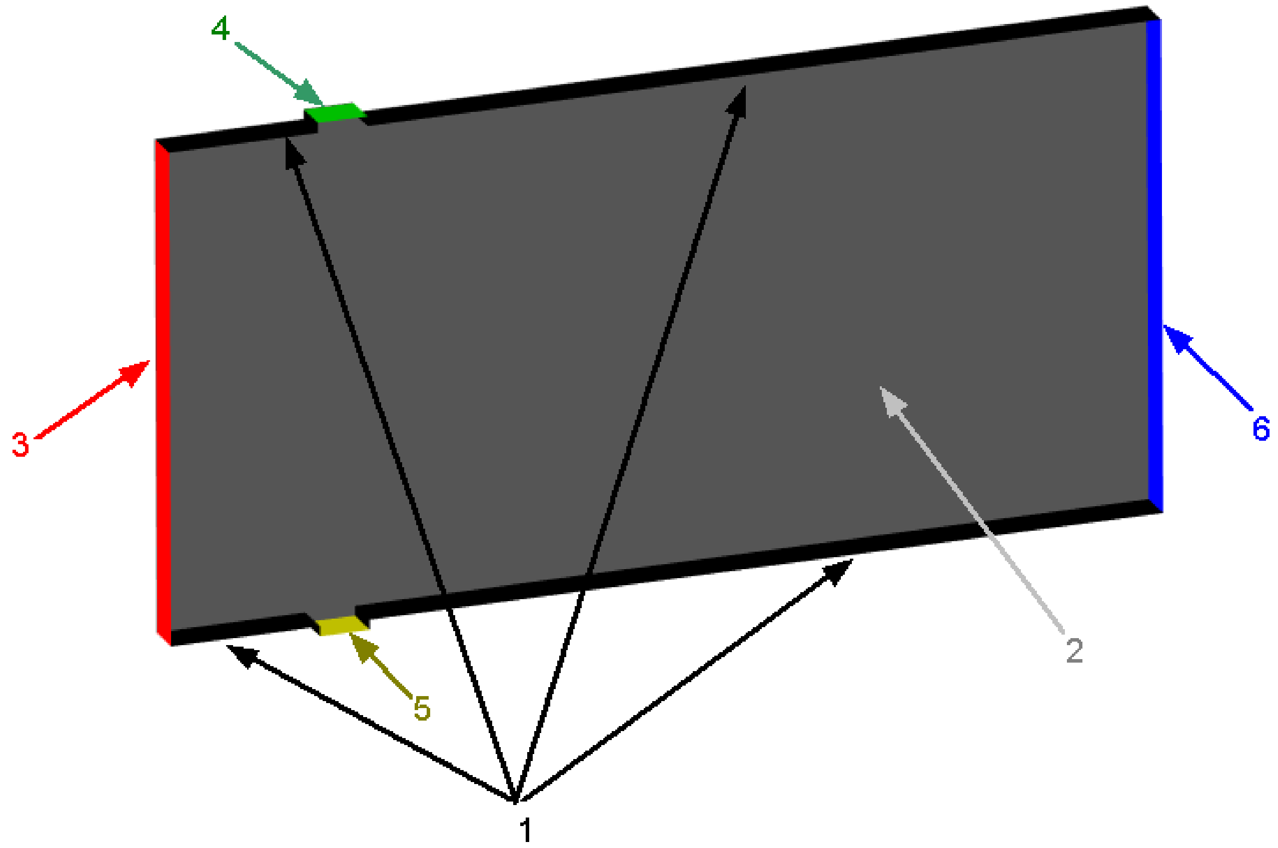
In the folder **Models**:

- Create **Model #0.**
- Add **Phase #0** into subfolder **Model #0 > Phases**

In the **Properties** window of the **SubRegion #0**, specify:

Model = Model #0

4.4.1.2 Boundary conditions



Specify the following boundary conditions:

Boundary 1

Type	**= Wall**
Variables	
Velocity(Phase #0)	**= Logarithm law**
Mass frac._ [Oxygen_Gas (equilibrium)] (Phase #0)	**= Zero gradient**
Mass frac._ [Nitrogen_Gas] (Phase #0)	**= Zero gradient**
TurbEnergy (Phase #0)	**= Value in cell near wall**
TurbDissipation (Phase #0)	**= Value in cell near wall**

Boundary 2

Type	**= Symmetry**
Variables	
Velocity(Phase #0)	**= Slip**
Mass frac._ [Oxygen_Gas (equilibrium)] (Phase #0)	**= Symmetry**
Mass frac._ [Nitrogen_Gas] (Phase #0)	**= Symmetry**
TurbEnergy (Phase #0)	**= Symmetry**
TurbDissipation (Phase #0)	**= Symmetry**

Boundary 3

Type	**= Inlet/Outlet**
Variables	

Velocity (Phase #0)	**= Normal mass velocity**	
Mass velocity	**= 0.129**	[kg m^{-2} s^{-1}]
Mass frac._ [Oxygen_Gas (equilibrium)] (Phase #0)	**= Value at the inlet**	
Value	**= 0**	
Mass frac._ [Nitrogen_Gas] (Phase #0)	**= Value at the inlet**	
Value	**= 0**	
TurbEnergy (Phase #0)	**= Pulsations**	
Value	**= 0**	
TurbDissipation (Phase #0)	**= Turbulent scale**	
Value	**= 0**	[m]

Boundary 4

Type	**= Inlet/Outlet**	
Variables		
Velocity (Phase #0)	**= Normal mass velocity**	
Mass velocity	**= 1.4**	[kg m^{-2} s^{-1}]
Mass frac. [Oxygen_Gas (equilibrium)] (Phase #0)	**= Value at the inlet**	
Value	**= 1**	
Mass frac._ [Nitrogen_Gas] (Phase #0)	**= Value at the inlet**	
Value	**= 0**	
TurbEnergy (Phase #0)	**= Pulsations**	
Value	**= 0**	
TurbDissipation (Phase #0)	**= Turbulent scale**	
Value	**= 0**	[m]

Boundary 5

Type	**= Inlet/Outlet**	
Variables		
Velocity (Phase #0)	**= Normal mass velocity**	
Mass velocity	**= 1.24**	[kg m^{-2} s^{-1}]
Mass frac. [Oxygen_Gas (equilibrium)] (Phase #0)	**= Value at the inlet**	
Value	**= 0**	
Mass frac. [Nitrogen_Gas] (Phase #0)	**= Value at the inlet**	
Value	**= 1**	
TurbEnergy (Phase #0)	**= Pulsations**	
Value	**= 0**	
TurbDissipation (Phase #0)	**= Turbulent scale**	
Value	**= 0**	[m]

Boundary 6

Type	**= Free Outlet**	
Variables		

Velocity (Phase #0)	**= Pressure**	
Value	**= 0**	[Pa]
Mass frac. [Oxygen_Gas (equilibrium)] (Phase #0)	**= Zero gradient**	
Mass frac. [Nitrogen_Gas] (Phase #0)	**= Zero gradient**	
TurbEnergy (Phase #0)	**= Zero gradient**	
TurbDissipation (Phase #0)	**= Zero gradient**	

4.4.1.3 Initial grid

Specify in the **Properties** window of the **Initial grid**:

Grid structure = 2D

Plane = XY

nX = 200

nY = 100

In the **Properties** window of the **Initial grid** click **Apply**.

4.4.1.4 Parameters of calculation

Specify in the **Solver** tab in properties of the **Time step** element:

Method	**= In seconds**	
Constant step	**= 0.1**	[s]

4.4.1.5 Visualization

To view the dynamics of the solution during the computation, specify visualization of **Concentration distribution** in the plane of the flow before the start of computation.

4.4.1.5.1 Concentration distribution

- In the **Properties** window of **Plane #0** specify:

Object

Normal

X	**= 0**
Y	**= 0**
Z	**= 1**

(to direct a **Plane**'s normal along the axis **Z**, you can also click the **Operations >**  button in the **Properties** window of the **Plane**)

- Create a layer **Color contours** on **Plane #0**.
- In the **Properties** window of the new **Color contours #0** layer specify:

Variable

Category	**= Variables of phase "Phase #0"**
Variable	**= Mass. frac. [Air_Gas (equilibrium)]**

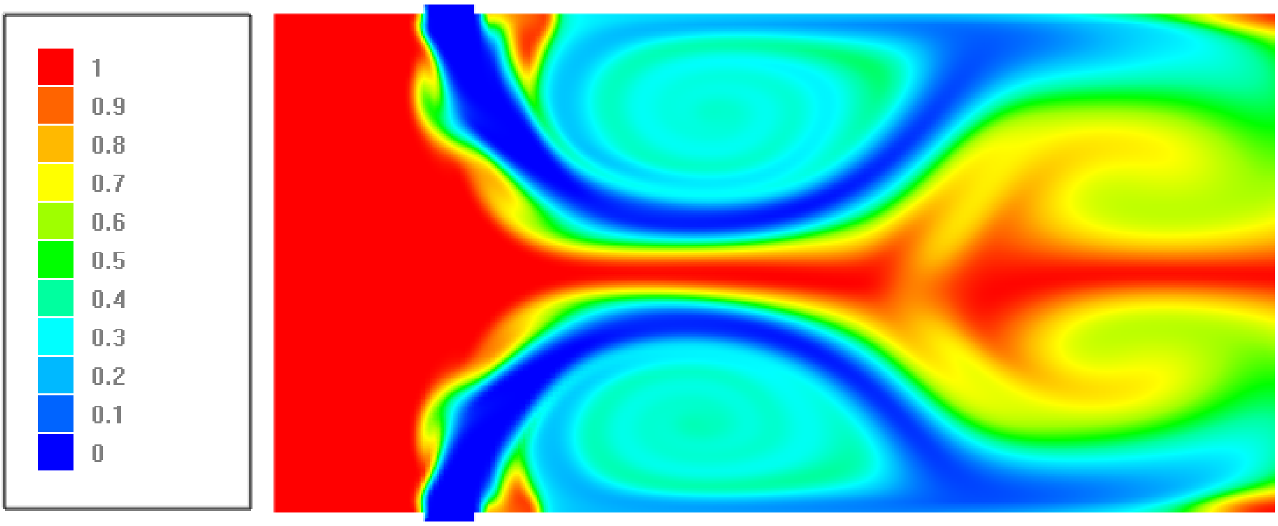
Value range

Mode	**= Manual**
Max	**= 1**

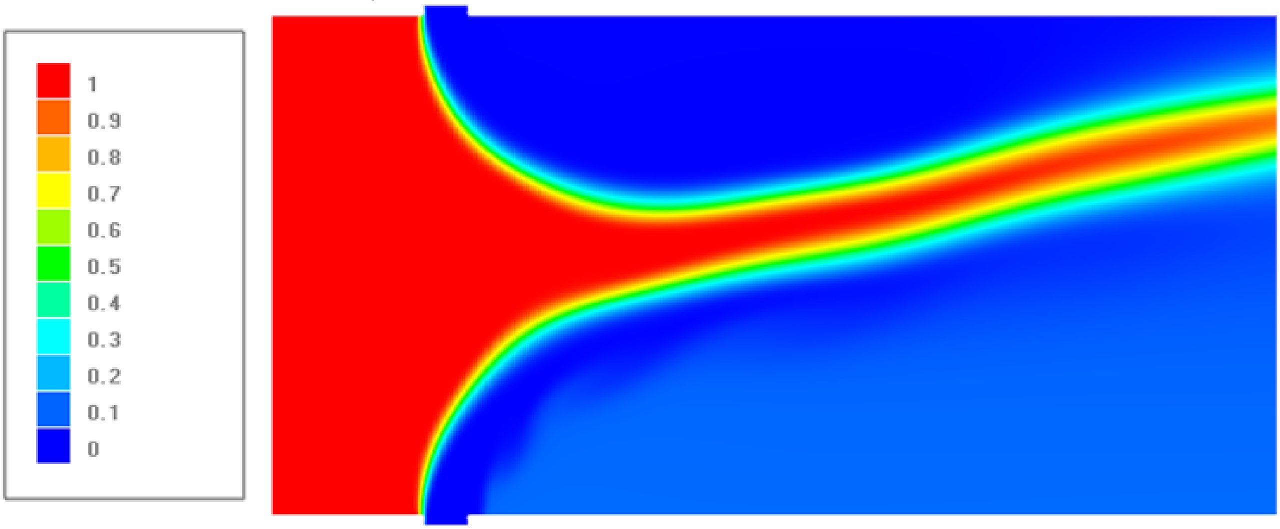
Min = 0

To display the palette, which is used for coloring the layer, you can use the **Info** window or, in properties of the layer, set **Appearance > Enabled = Yes**. To open the **Info** window, you have to select the layer in the tree of **Postprocessor** and click the button.

At the beginning of the computation, the **Color contours #0** layer will look as shown below:



After 1000 iterations the stream pattern will settle as shown on the illustration below:



4.4.2 Radioactive decay

Consider the simulation of radioactive decay of an isotope.

Parameters of the problem setting

The length of the area:	L	= 0.7	[m]
Inflow parameters:			
Flow velocity:	V	= 1	[m s^{-1}]

Mass fraction of isotopes:	Y	= 10^{-3}	

Properties of sodium with the isotope:

Molar mass	M	= 0.023	[kg $mole^{-1}$]
Density	ρ	= 925	[kg m^{-3}]
Viscosity	μ	= 6.68×10^{-4}	[kg m^{-1} s^{-1}]
Thermal conductivity	λ	= 84.9	[W m^{-1} K^{-1}]
Specific heat	Cp	= 1382	[J kg^{-1} K^{-1}]

Geometry file and project's name

Geometry: `Isotope.wrl`

Project: `Isotope`

4.4.2.1 Physical model

In properties of the element **General settings** specify:

Reference values

Temperature	**= 373**	[K]

In the folder **Substances**:

- Create **Substance #0**.
- In properties of **Substance #0** specify:

Name	**= Sodium**	
Aggregative state	**= Liquid**	
Molar mass		
Value	**= 0.023**	[kg $mole^{-1}$]
Density		
Value	**=925**	[kg m^{-3}]
Viscosity		
Value	**= 6.68e-4**	[kg $m^{-1}s^{-1}$]
Thermal conductivity		
Value	**= 84.9**	[W $m^{-1}K^{-1}$]
Specific heat		
Value	**= 1382**	[J kg^{-1} K^{-1}]
Enthalpy of formation		
Value	**= 0**	[J kg^{-1}]

- Copy **Sodium**.

- In properties of the new **Substance** specify:

Name	**= Isotope**	
Enthalpy of formation		
Value	= **1e6**	[J kg^{-1}]

In the folder **Phases**:

- Create a continuous **Phase #0**.
- In **Phase #0** add both the just created substances into the folder **Substances** in the following order:
  - **Isotope**
  - **Sodium**

  Sodium is placed on the second place because its mass fraction in the **Phase** is greater.
- In properties of of the folder **Phase #0 > Physical processes** specify:

Heat transfer	= **Heat transfer via h**
Motion	= **Navier-Stokes model**
Mass transfer	= **Mixing**

In properties of the element **Phases > Phase #0 > Physical processes > Mass transfer > Substances > Isotope** specify:

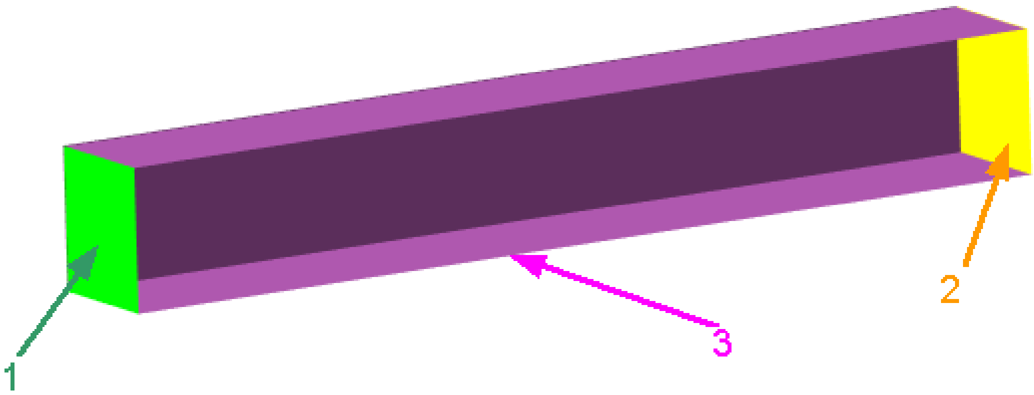
D	= **-925**

In the folder **Models**:

- Create **Model #0**.
- Add **Phase #0** into the subfolder **Model #0 > Phases**.

In properties of **SubRegion #0**, specify **Model = Model #0**.

4.4.2.2 Boundary conditions



Specify boundary conditions:

Boundary 1

Type	= **Inlet/Outlet**	
Variables		
Temperature (Phase #0)	= **Temperature**	
Value	= **0**	[K]
Velocity (Phase #0)	= **Normal mass velocity**	
Mass velocity	= **925**	[kg m^{-2} s^{-1}]
Mass frac. [Isotope] (Phase #0)	= **Value at the inlet**	
Value	= **1e-3**	

Boundary 2

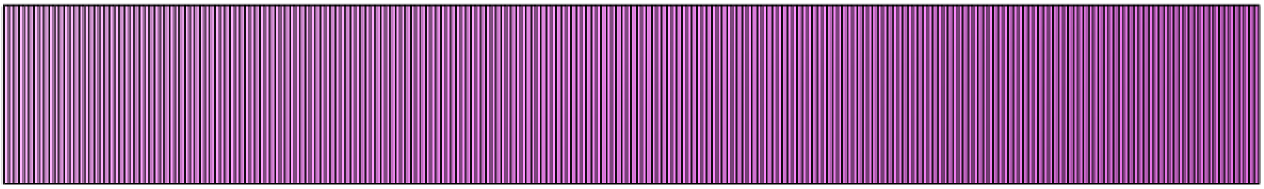
Type	= **Free Outlet**
Variables	

Temperature (Phase #0)	**= Zero gradient**	
Velocity (Phase #0)	**= Pressure**	
Value	**= 0**	[Pa]
Mass frac._ [Isotope] (Phase #0)	**= Zero gradient**	

Boundary 3

Type	**= Symmetry**
Variables	
Temperature (Phase #0)	**= Symmetry**
Velocity (Phase #0)	**= Slip**
Mass frac._ [Isotope] (Phase #0)	**= Symmetry**

4.4.2.3 Initial grid



In properties of the **Initial grid** specify:

Grid structure = 1D
Direction = X
nX = 250

In the **Properties** window of the **Initial grid** click **Apply**.

4.4.2.4 Parameters of calculation

In the **Solver** tab in properties of the **Time step** element specify:

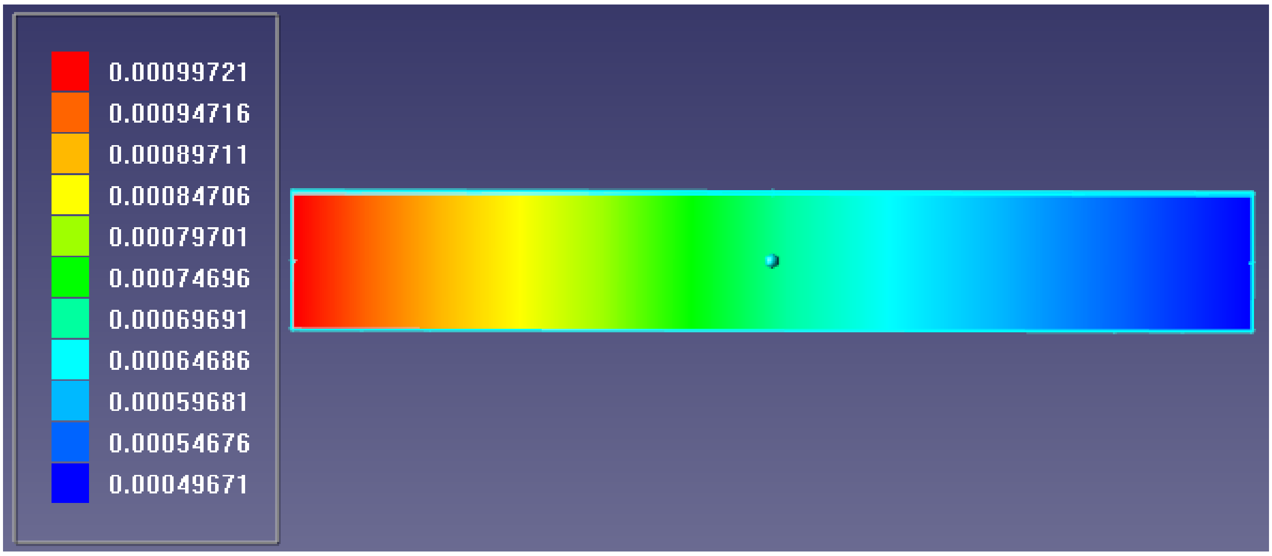
Method	**= Via CFL number**	
Convective CFL	**= 1**	
Diffusive CFL	**= 1**	
Max step	**= 0.01**	[s]

4.4.2.5 Visualization

Specify visualization of:

1. **Isotope concentration distribution** in the plane of the flow
2. **Temperature distribution** in the plane of the flow

4.4.2.5.1 Isotope concentration distribution



- In the **Properties** window of **Plane #0** specify:

Object

Normal		
	X	**= 0**
	Y	**= 0**
	Z	**= 1**

(to direct a **Plane**'s normal along the axis **Z**, you can also click the **Operations >** button in the **Properties** window of the **Plane**)

- Create the layer **Color contours #0** on **Plane #0**.
- In properties of **Color contours #0** specify:

Variable

Category	= **Variables of phase "Phase #0"**
Variable	**= Mass. frac. [Isotope]**

Palette

Appearance		
	Enabled	**= Yes**
	Style	**= Style 3**

4.4.2.5.2 Temperature distribution

- Create a **Color contours** layer in **Plane #0**.
- In the **Properties** window of the **Color contours** specify:

Variable

	Variable	**= Temperature**

Value range

	Mode	**= Manual**
	Max	**= 0.35**
	Min	**= 0**

Palette

Appearance

	Enabled	**= Yes**
	Style	**= Style 1**

4.4.3 Chemistry (Dissociation of Nitrogen)

In this example we simulate dissociation of nitrogen, which includes two reversible chemical reactions:

1. $N_2 + N_2 \leftrightarrow 2N + N_2$
2. $N_2 + N \leftrightarrow 2N + N$

Parameters of the problem setting

The length of the area:	= 0.7	[m]
Inflow parameters:		
Flow velocity:	= 14	[m s^{-1}]
Mass fraction of N_2 on inlet:	= 100%	

Geometry file and project's name:

Geometry: `Isotope.wrl`

Project: **Dissociation_N2**

4.4.3.1 Physical model

Create a project based on the geometry file `Isotope.wrl` (this file used before for the exercise Radioactive decay).

In the **Properties** window of the element **General settings** specify:

Reference values

Temperature	**= 300**	[K]
Pressure	**= 101325**	[Pa]

In the folder **Substances**:

- Create **Substance #0.**
- Specify the following properties of **Substance #0**:

Name	**= N2**	
Aggregative state	**= Gas**	
Molar mass		
Value	**= 0.028**	[kg mole^{-1}]
Density		
Value	**= Ideal gas law**	[kg m^{-3}]
Specific heat		
Value	**= 1039.3**	[J kg^{-1} K^{-1}]
Enthalpy of formation		
Value	**= 0**	[J kg^{-1}]

- Create another **Substance** and specify its properties:

Name	**= N**	
Aggregative state	**= Gas**	
Molar mass		
Value	**= 0.014**	[kg mole^{-1}]
Density		
Value	**= Ideal gas law**	[kg m^{-3}]
Specific heat		
Value	**= 1484.7**	[J kg^{-1} K^{-1}]
Enthalpy of formation		
Value	**= 33762857.1**	[J kg^{-1}]

In the folder **Phases**:

- Create a continuous **Phase #0.**
- In **Phase #0** add both the just created substances (**N2** and **N**) into the folder **Substances.**
- In the **Properties** window of the folder **Phase #0** > **Physical processes** specify:

Heat transfer	**= Heat transfer via h**
Motion	**= Navier-Stokes model**
Mass transfer	**= Chemistry**

- In the folder **Phases > Phase #0 > Physical processes > Mass transfer > Reactions** create **Reaction #0** using the **Create** command of the command menu.
- In the **Properties** window of the just created **Reaction #0** specify*):

Af	**= 192000000000**
nf	**= -0.5**
Tf	**= 113100**
Ar	**= 10900**
nr	**= -0.5**
Tr	**= 0**
Stoichiometric coeffs. > N > Real	**= 2**
Stoichiometric coeffs. > N > Effective	**= 2**
Stoichiometric coeffs. > N2 > Real	**= -1**
Stoichiometric coeffs. > N2 > Effective	**= -1**
Efficiencies > N	**= 0**
Efficiencies > N2	**= 2.5**

*) constants of the forward and reverse reactions $N_2 + N_2 \leftrightarrow 2N + N_2$ are defined by the following expressions:

$$k_{f,1} = 1.92{\cdot}10^{11} T^{-0.5} e^{-113100/T}$$

$$k_{r,1} = 1.09{\cdot}10^{4} T^{-0.5}$$

Reaction #0 ($N_2 + N_2 \leftrightarrow 2N + N_2$) has the following *molar* formula:

$$N_2 \leftrightarrow 2N$$

According to this formula, the real stoichiometric coefficients are specified as follows:

- **-1** for molecular nitrogen N_2
- **2** for atomic nitrogen N

For **Substances** from the left side of the molar reaction formula, stoichiometric coefficients are specified as negative values. For **Substances** from the right side of the formula, stoichiometric coefficients are specified as positive values.

The simulated chemical reactions are elementary, so effective stoichiometric coefficients are the same as the real ones.

- In the folder **Phases > Phase #0 > Physical processes > Mass transfer > Reactions** create another chemical reaction, **Reaction #1**.
- In properties of **Reaction #1** specify**):

Af	**= 4.15e+016**
nf	**= -1.5**
Tf	**= 113100**
Ar	**= 2320000000**
nr	**= -1.5**
Tr	**= 0**
Stoichiometric coeffs. > N > Real	**= 2**
Stoichiometric coeffs. > N > Effective	**= 2**

Stoichiometric coeffs. > N2 > Real	**= -1**
Stoichiometric coeffs. > N2 > Effective	**= -1**
Efficiencies > N	**= 1**
Efficiencies > N2	**= 0**

**) constants of the forward and reverse reactions $N_2 + N \leftrightarrow 2N + N$ are defined by the following expressions:

$k_{f,2} = 4.15 \cdot 10^{16} T^{-1.5} e^{-113100/T}$

$k_{r,2} = 2.32 \cdot 10^{9} T^{-1.5}$

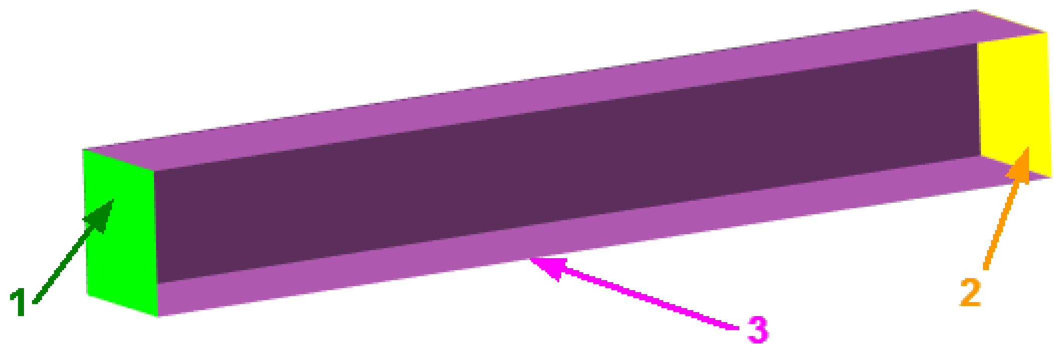
Reaction #1 ($N_2 + N \leftrightarrow 2N + N$) has the same molar formula, $N_2 \leftrightarrow 2N$.
So you have to specify for it the same stoichiometric coefficients.

In the folder **Models**:

- Create **Model #0**.
- Add **Phase #0** into the subfolder **Model #0 > Phases**.

In the **Properties** window of **SubRegion #0**, specify **Model = Model #0**.

4.4.3.2 Boundary conditions



Specify boundary conditions:

Boundary 1

Type	**= Inlet/Outlet**	
Variables		
Temperature (Phase #0)	**= Temperature**	
Value	**= 4700**	[K]
Velocity (Phase #0)	**= Normal mass velocity**	
Mass velocity	**= 1**	[kg m^{-2} s^{-1}]
Mass frac. [N2](Phase #0)	**= Value at the inlet**	
Value	= **1**	
Mass frac. [N](Phase #0)	**= Value at the inlet**	
Value	**= 0**	

Boundary 2

Type	**= Free Outlet**
Variables	

Temperature (Phase #0)	**= Zero gradient**	
Velocity (Phase #0)	**= Pressure**	
Value	**= 0**	[Pa]
Mass frac. [N2](Phase #0)	**= Zero gradient**	
Mass frac. [N](Phase #0)	**= Zero gradient**	

Boundary 3

Type	**= Symmetry**
Variables	
Temperature (Phase #0)	**= Symmetry**
Velocity (Phase #0)	**= Slip**
Mass frac. [N2](Phase #0)	**= Symmetry**
Mass frac. [N](Phase #0)	**= Symmetry**

4.4.3.3 Initial conditions

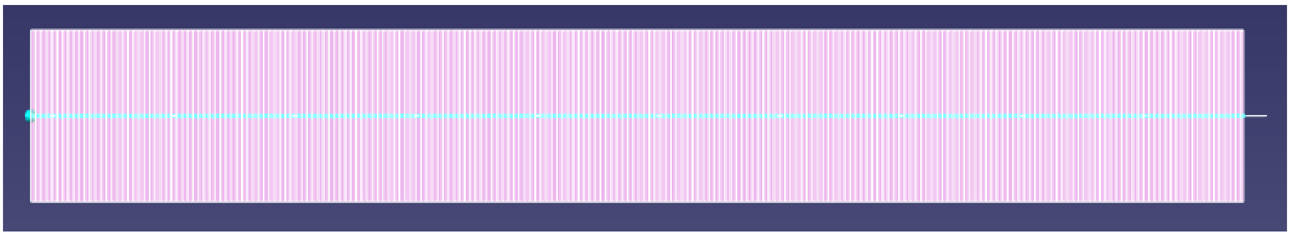
In properties of child elements in the subfolder **Models > Model #0 > Init. data > Init. data #0** specify:

- for the element **Temperature (Phase #0)** specify **Value = 4700**.
- for the element **Velocity (Phase #0)** specify **Value > X = 14**, **Value > Y = 0**, **Value > Z = 0**.
- for the element **Pressure (Phase #0)** specify **Value = 0**.
- for the element **Mass frac. [N2] (Phase #0)** specify **Value = 1**.
- for the element **Mass frac. [N] (Phase #0)** specify **Value = 0**.

In properties of the element **SubRegion #0 > Initial conditions > Init. condition #0** specify:

Object	**= Computational space**
Init. data	**= Init. data #0**

4.4.3.4 Initial grid



In properties of the **Initial grid** specify:
Grid structure = 1D
Direction = X
nX = 250

In the **Properties** window of the **Initial grid** click **Apply**.

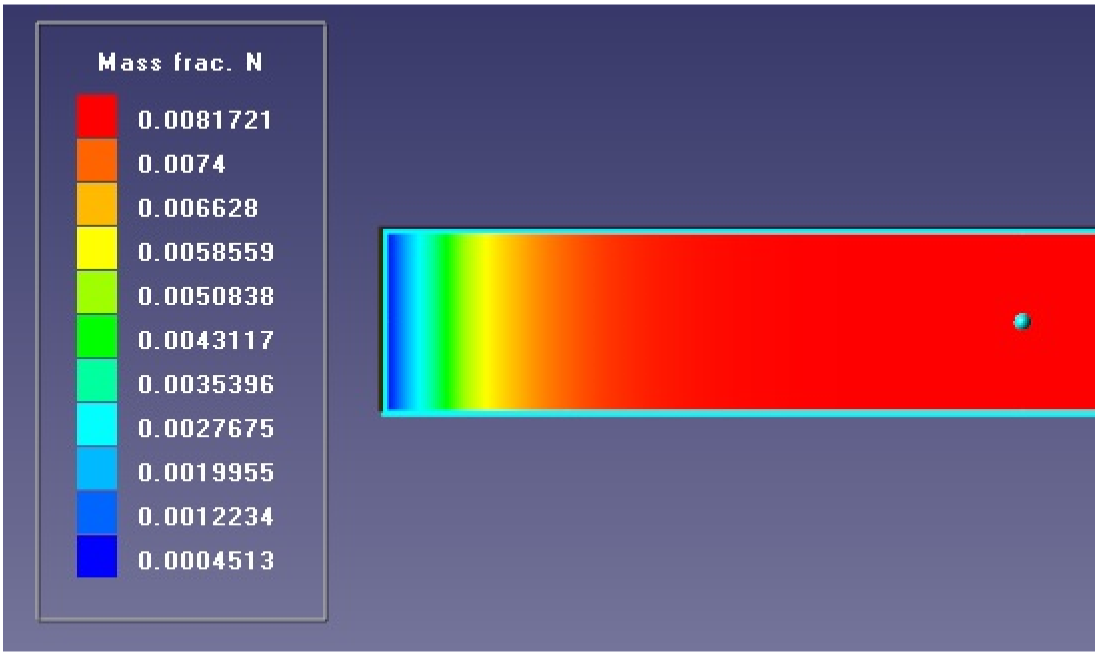
4.4.3.5 Parameters of calculation

In the **Solver** tab in properties of the **Time step** element specify:

Method	**= Via CFL number**
Convective CFL	**= 5**
Diffusive CFL	**= 5**

Max step **= 1** [s]

4.4.3.6 Visualization



Create a **Layer**, which will display distribution of mass fraction of dissociated nitrogen:

- In properties of **Plane #0** specify:

Object

Normal

X	**= 0**
Y	**= 0**
Z	**= 1**

(to direct a **Plane**'s normal along the axis **Z**, you can also click the **Operations >** button in the **Properties** window of the **Plane**)

- Create the layer **Color contours #0** on **Plane #0**.
- In properties of **Color contours #0** specify:

Name	**= Mass frac. N**
Variable	
Category	**= Variables of phase "Phase #0"**
Variable	**= Mass frac. [N]**
Palette	
Appearance	
Enabled	**= Yes**
Title	**= Yes**
Style	**= Style 3**

Run the calculation of the project and view the distribution of fraction of dissociated nitrogen.

4.4.4 Combustion

Consider simulating the combustion of natural gas in the air.

Parameters of the problem setting

Length of the tube:	L	= 0.1	[m]
Substances			
Fuel		= Natural Gas	
Oxidant		= Air	
Inflow parameters:			
Speed of natural gas:	V	= 75	[m s^{-1}]
Pressure:	P	= 101 000	[Pa]
Mixture combustion parameters:			
Steheometrichesky ratio of oxidant	i_1	= 16.92	
Steheometrichesky ratio of combustion products	i_2	= 17.92	
Flashpoint	T	= 923	[K]
Alpha min.	α_min	= 0.6	
Alpha max.	α_max	= 1.9	

Geometry file and project's name

Geometry: `Combustion.wrl`

Project: `Combustion`

4.4.4.1 Physical model

In the **Properties** window of the element **General settings** specify:

Reference values

Temperature	**= 298**	[K]
Pressure	**= 101000**	[Pa]

In the folder **Substances**:

- Create **Substance #0**.
- Load **Substance #0** from the **Standard** substance database:

- From the context menu of **Substance #0** select **Load from SD > Standard.**
- In the new window **Load from database**, select:

Substances	**= Natural Gas**
Phases	**= Gas (equilibrium)**

- Create**Substance #0**.
- Load **Substance #0** from the **Standard** substance database:
  - From the context menu of **Substance #0** select **Load from SD > Standard.**
  - In the new window **Load from database**, select:

Substances	**= Air**
Phases	**= Gas**

- Create**Substance #0**.
- Load **Substance #0** from the **Standard** substance database:
  - From the context menu of **Substance #0** select **Load from SD > Standard.**
  - In the new window **Load from database**, select:

Substances	**= Natural gas+Air, products**
Phases	**= Gas (equilibrium)**

In the folder **Phases**:

- Create a continuous **Phase #0.**
- In **Phase #0**, add into the folder **Substances** all the just created substances:
  - **Natural gas_Gas (equilibrium)**
  - **Air_Gas**
  - **Natural gas+Air, products_Gas (equilibrium)**
- Specify properties of the folder **Phase #0 > Physical processes**:

Heat transfer	**= Heat transfer via H**
Motion	**= Navier-Stokes model**
Mass transfer	**= Combustion**
Turbulence	**= KES**

In **Phase #0 > Physical Processes > Mass transfer** specify:

Combustion model	**= Arrhenius-Magnussen**	
i_1	**= 16.92**	
i_2	**= 17.92**	
T ignition	**= 923**	[K]
Alpha min.	**= 0.6**	
Alpha max.	**= 1.9**	
Fuel	**= Natural gas_Gas (equilibrium)**	
Oxidizer	**= Air_Gas**	
Product-1	**= Natural gas+Air, products_Gas (equilibrium)**	

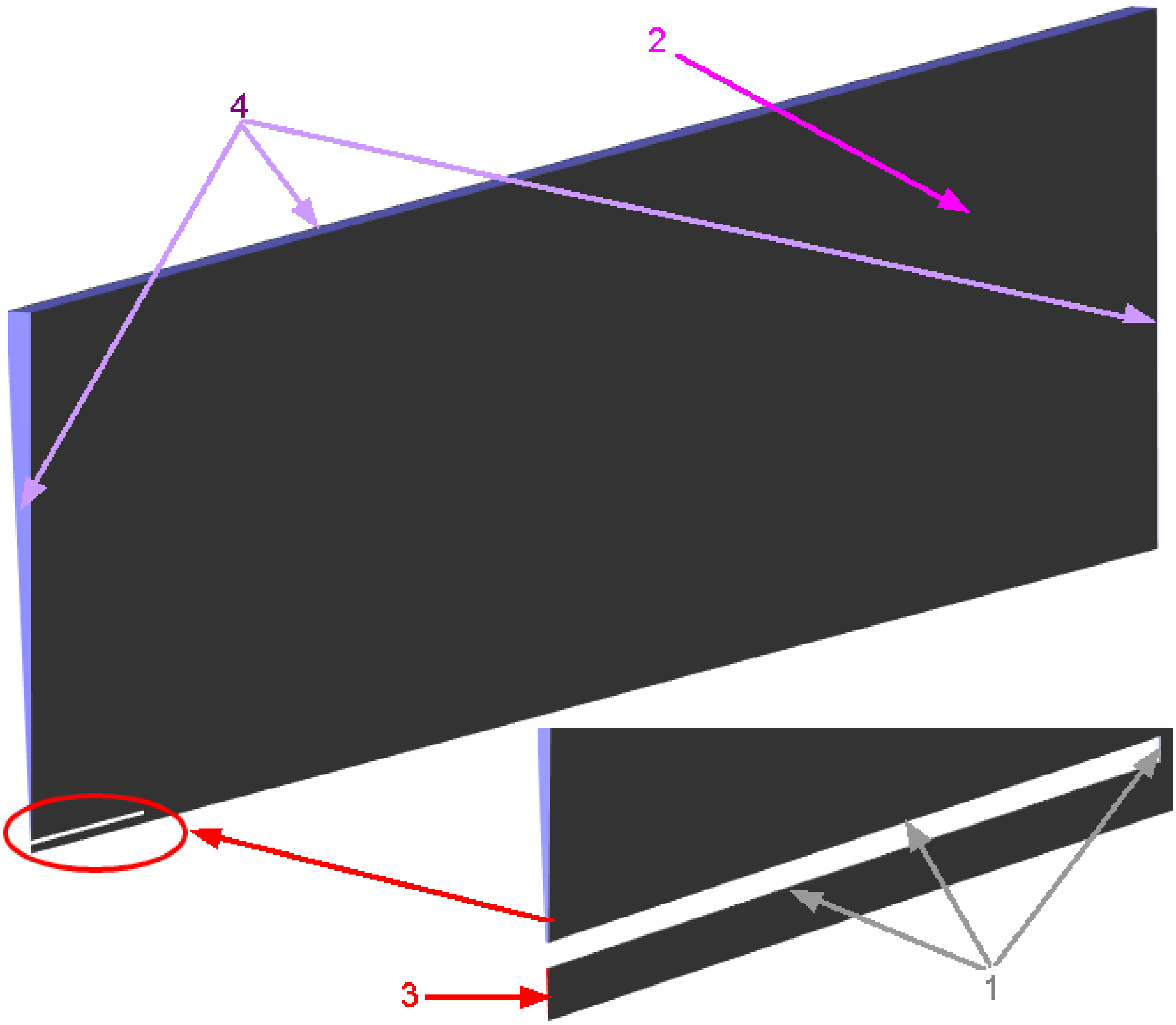
In the folder **Models**:

- Create **Model #0**.
- Add **Phase #0** into subfolder **Model #0 > Phases**

In the **Properties** window of the **SubRegion #0**, specify:

Model = Model #0

4.4.4.2 Boundary conditions



Specify the following boundary conditions:

Boundary 1

Type	**= Wall**
Variables	
Temperature(Phase #0)	**= Zero gradient**
Velocity(Phase #0)	**= Logarithm law**
Mass frac._[Natural gas_Gas (equilibrium)](Phase #0)	**= Zero gradient**
Mass frac._[Natural gas+Air,products_Gas (equilibrium)](Phase #0)	**= Zero gradient**
Variance of fuel(Phase #0)	**= Zero gradient**
TurbEnergy(Phase #0)	**= Value in cell near wall**
TurbDissipation(Phase #0)	**= Value in cell near wall**

Boundary 2

Type	**= Symmetry**	
Variables		
Temperature(Phase #0)	**= Symmetry**	
Velocity(Phase #0)	**= Slip**	
Mass frac._[Natural gas_Gas (equilibrium)](Phase #0)	**= Symmetry**	
Mass frac._[Natural gas+Air,products_Gas (equilibrium)](Phase #0)	**= Symmetry**	
Variance of fuel(Phase #0)	**= Symmetry**	
TurbEnergy (Phase #0)	**= Symmetry**	
TurbDissipation (Phase #0)	**= Symmetry**	

Boundary 3

Type	**= Inlet/Outlet**	
Variables		
Temperature(Phase #0)	**= Temperature**	
Value	**= 0**	[K]
Velocity(Phase #0)	**= Normal mass velocity**	
Mass velocity	**= 50**	[kg $m^{-2}s^{-1}$]
Mass frac._[Natural gas_Gas (equilibrium)](Phase #0)	**=Value at the inlet**	
Value	**= 1**	
Mass frac._[Natural gas+Air,products_Gas (equilibrium)] (Phase #0)	**=Value at the inlet**	
Value	**= 0**	
Variance of fuel(Phase #0)	**=Value at the inlet**	
Value	**= 0**	
TurbEnergy(Phase #0)	**= Pulsations**	
Value	**= 0.03**	
TurbDissipation(Phase #0)	**= Turbulent scale**	
Value	**= 0.0008**	[m]

Boundary 4

Type	**= Inlet/Outlet**	
Variables		
Temperature(Phase #0)	**= Temperature**	
Value	**= 0**	
Velocity(Phase #0)	**= Inlet pressure**	
Value	**= 0**	[kg $m^{-2}s^{-1}$]
Mass frac._[Natural gas_Gas (equilibrium)](Phase #0)	**= Value at the inlet**	
Value	**= 0**	
Mass frac._[Natural gas+Air,products_Gas (equilibrium)] (Phase #0)	**= Value at the inlet**	
Value	**= 0**	
Variance of fuel(Phase #0)	**= Value at the inlet**	
Value	**= 0**	
TurbEnergy(Phase #0)	**= Pulsations**	
Value	**= 0.03**	
TurbDissipation(Phase #0)	**= Turbulent scale**	
Value	**= 0.01**	[m]

4.4.4.3 Ignition

In this simulation, you have to burn the fuel in the area where the fuel is mixing with the oxidant.

To do this, you have to specify an **Ignition Modifier** located in this area.

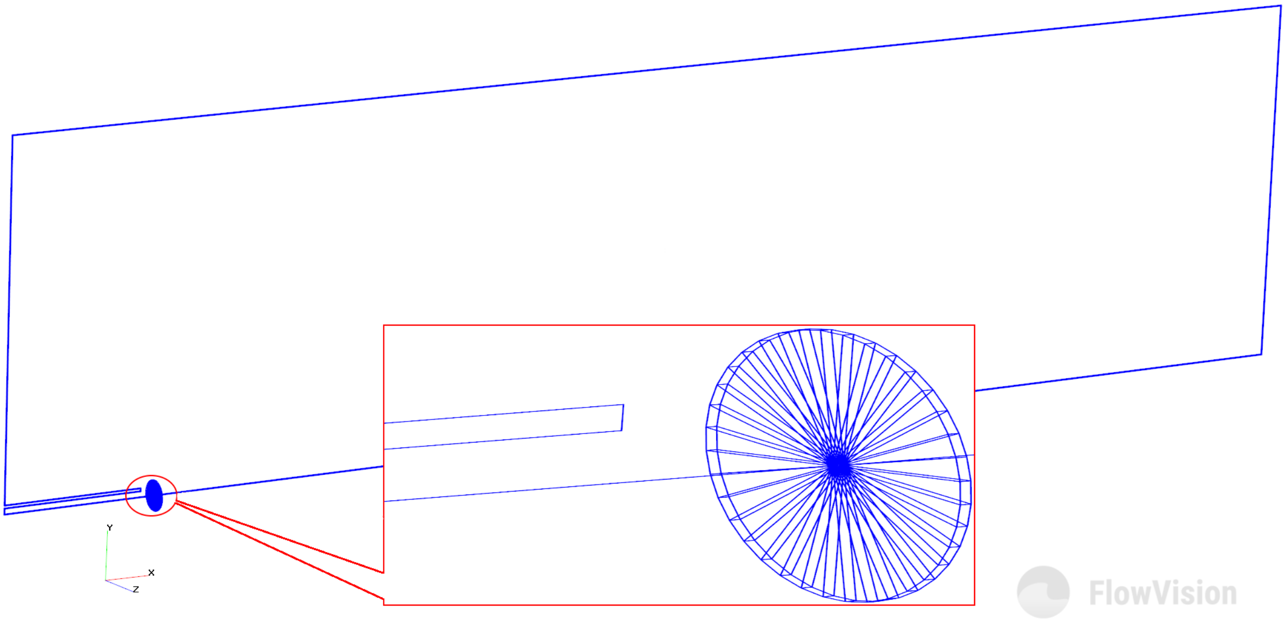
The procedure of setting a **Modifier** consists of two steps:

- specifying an area where the **Modifier** is to be active
- specifying the **Modifier** in this area

Specify the area where the **Modifier** is to be active:

- In the folder **Objects** create a **Cone/cylinder**.
- In the **Properties** window of the object **Cone/cylinder #0** specify:

Location		
Reference point		
X	**= 0.11**	[m]
Y	**= 0**	[m]
Z	**= 0**	[m]
Parameters		
Height	**= 0.0005**	[m]
Radius 1	**= 0.01**	[m]
Radius 2	**= 0.01**	[m]
Base ratio	**= 1**	
Channel	**= None**	



Create a **Modifier** for ignition:

- From the context menu of the folder **Modifiers**, select **Create**.
- In the **Create new modifier** window, specify:

Modifier type	= **Ignition / extinction zone**
Objects	= **Cone/cylinder #0**

- In the **Properties** window of the just created **Ignition / extinction zone #0** modifier specify:

Activation	
Type	**= Only once by step**

Start in steps = **10**

Duration in steps = **40**

Type = **Ignition**

Note:

It is recommended not to activate an **Ignition** at the beginning of the computation but only after the flow is formed.

After switching an **Ignition** off, the flame can temporarily extinguish.

4.4.4.4 Initial conditions

Specify parameters of the initial distribution of the oxidant:

- In properties of child elements in the folder **Models > Model #0 > Init. data > Init. data #0** specify:
  - for the element **Velocity (Phase #0)**:

X	**= 0.1**	[m s^{-1}]
Y	**= 0**	[m s^{-1}]
Z	**= 0**	[m s^{-1}]

  - for the element **Pulsations (Phase #0)**:

    Value **= 0.03**

  - for the element **Turbulent scale (Phase #0)**:

    Value **= 0.01** [m]

Specify the parameters for the initial distribution of fuel:

- In the folder **Models > Model #0 > Init. data** create **Init. data #1**.
- In properties of child elements of **Init. data #1** specify:
  - for the element **Velocity (Phase #0)**:

X	**= 75**	[m s^{-1}]
Y	**= 0**	[m s^{-1}]
Z	**= 0**	[m s^{-1}]

  - for the element **Mass frac._Natural gas_Gas (equilibrium)(Phase #0)**:

    Value **= 1**

  - for the element **Pulsations (Phase #0)**:

    Value **= 0.03**

  - for the element **Turbulent scale (Phase #0)**:

    Value **= 0.0008** [m]

- In the **Objects** folder create **Cone/cylinder #1**.
- In properties of **Cone/cylinder #1** specify:

  Parameters

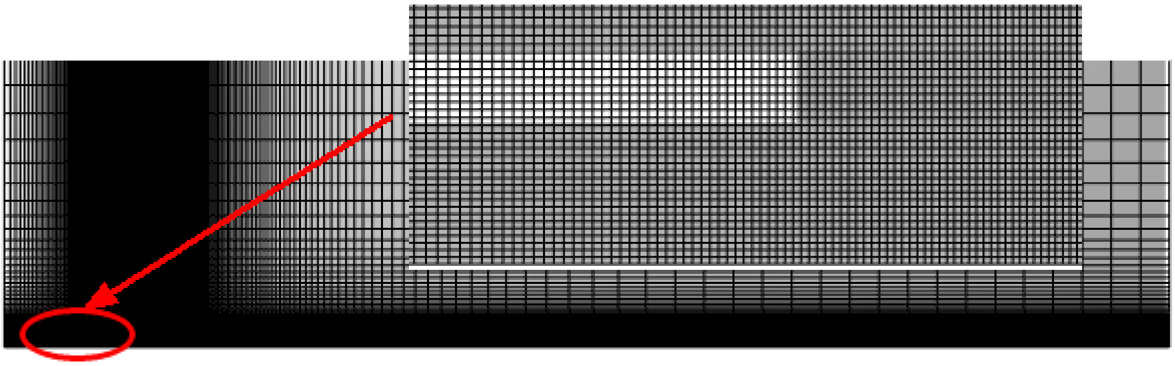
Height	**= 0.105**	[m]
Radius 1	**= 0.005**	[m]
Radius 2	**= 0.005**	[m]
Base ratio	**= 1**	

- In the folder **SubRegion #0 > Initial conditions** create the object **Init. condition #1**.
- In the **Properties** window of the object **Init. condition #1** specify:

  Object **= Cone/cylinder #1**

Init. data **= Init. data #1**

4.4.4.5 Initial grid



In the **Properties** window of the **Initial grid**, click the button  to open the **Initial grid editor**.

Specify in the **Initial grid editor**:

for axis OX

Grid parameters

h_max **= 0.025** [m]

h_min **= 0.0005** [m]

Insert a reference line with coordinate **x=0.11** [m].

Specify **Reference line parameters** for the reference line with coordinate **x=0**:

h **= 0.005** [m]

kh+ **= 0.9**

Specify **Reference line parameters** for the reference line with coordinate **x=0.11**:

h **= 0.0005** [m]

kh- **= 1**

kh+ **= 1**

Specify **Reference line parameters** for the reference line with coordinate **x=1**:

h **= 0.025** [m]

kh- **= 0.9**

for axis OY (click the button )

Grid parameters

h_max **= 0.025** [m]

h_min **= 0.0005** [m]

Specify **Reference line parameters** for the reference line with coordinate **y=0**:

h **= 0.0005** [m]

kh+ **= 1**

Specify **Reference line parameters** for the reference line with coordinate **y=0.25**:

h **= 0.025** [m]

kh- **= 1**

Click **OK** to close the **Initial grid editor** with saving the entered data.

Specify in the **Properties** window of the **Initial grid**:

Grid structure = 2D

Plane = XY

In the **Properties** window of the **Initial grid** click **Apply**.

In the **Postprocessor** tab, in properties of the element **Layers > Initial grid** specify **Lines > Color = Black**.

4.4.4.6 Parameters of calculation

Specify in the **Solver** tab:

- In properties of the **Time step** element specify:

Method	**= Via CFL number**	
Convective CFL	**= 500**	
Max step	**= 0.001**	[s]

- In properties of the **Limiters > Limiters for calculation > Phase limiters > Phase #0** element specify:

Limiter

Density, min.	**= 0.01**	[kg m^{-3}]
Temperature abs, min.	**= 200**	[K]
Temperature abs, max.	**= 3000**	[K]
Velocity, max.	**= 1000**	[m s^{-1}]
Pressure abs, min.	**= 1000**	[Pa]
Pressure abs, max.	**= 300000**	[Pa]

- In properties of the **Stopping conditions > Time steps** element specify **Number = 50**.

4.4.4.7 Preliminary and the main calculations

On the **Solver** tab in properties of the element **Advanced settings** specify:

Numerical method > Type of scheme	**= Implicit**
Numerical method > Pressure gradient	**= Simple**

Start the project's computation.

After 50 steps (when the ignition finishes its activity) stop the computation and on the **Solver** tab in properties of the **Time step** element specify:

Method	**= Via CFL number**
Convective CFL	**= 10**

(Value of the **Max step** parameter is not changed)

To obtain a stable flow it is necessary to run the computation with a large number of steps.

- On the **Solver** tab in properties of the **Stopping conditions > Time steps** element specify **Number = 12000**.
- Continue the computation.

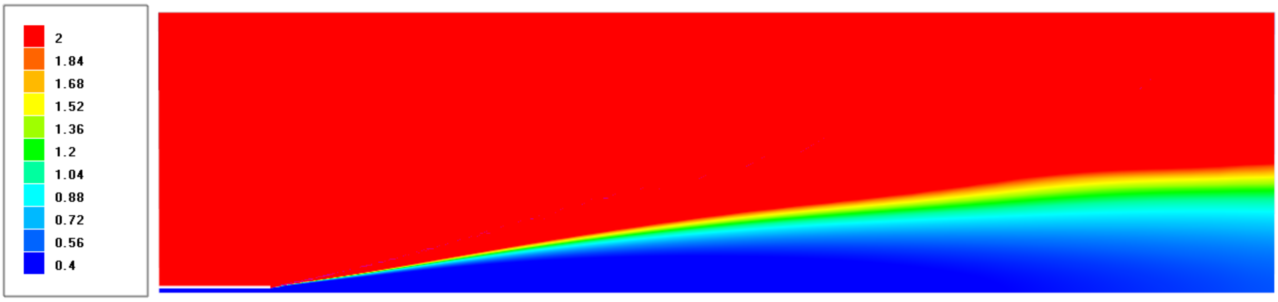
4.4.4.8 Visualization

To view the dynamics of the solution during the computation, prior to computation, specify visualization of:

1. Oxidant excess factor's distribution in the plane of the flow.
2. Temperature distribution in the plane of the flow.

4.4.4.8.1 Oxidant excess factor's distribution

Visualization at the step number 12000:



- In the **Properties** window of **Plane #0** specify:

Object

Normal

X	**= 0**
Y	**= 0**
Z	**= 1**

(to direct a **Plane**'s normal along the axis **Z**, you can also click the **Operations >** button in the **Properties** window of the **Plane**)

- Create a layer **Color contours** on **Plane #0.**
- In the **Properties** window of the **Color contours #0** specify:

Variable

Category	**= Variables of phase "Phase #0"**
Variable	**= Oxid excess factor rec.**

Value range

Mode	**= Manual**
Max	**= 2**
Min	**= 0.4**

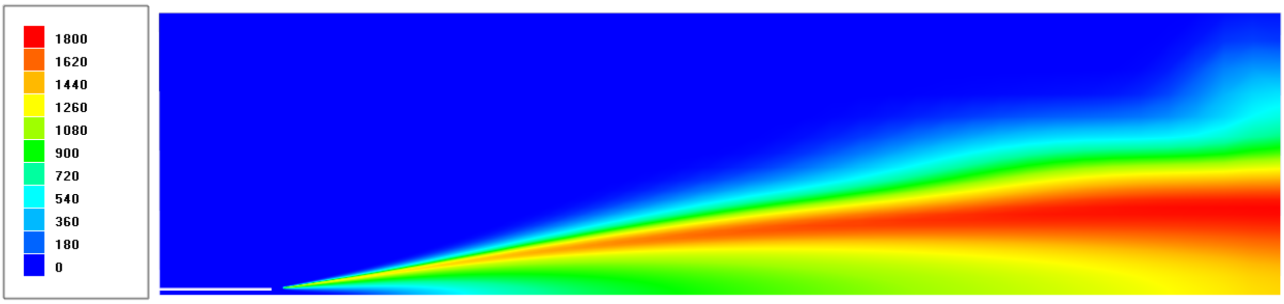
Palette

Appearance

Enabled	**= Yes**

4.4.4.8.2 Temperature distribution

Visualization at the step number 12000:



- Create a **Color contours** layer on **Plane #0.**
- In the **Properties** window of the layer **Color contours #1** specify:

Variable

Variable	**= Temperature**

Value range

Mode	= Manual
Max	= 1800
Min	= 0
Palette	
Appearance	
Enabled	= Yes

4.5 Free surface

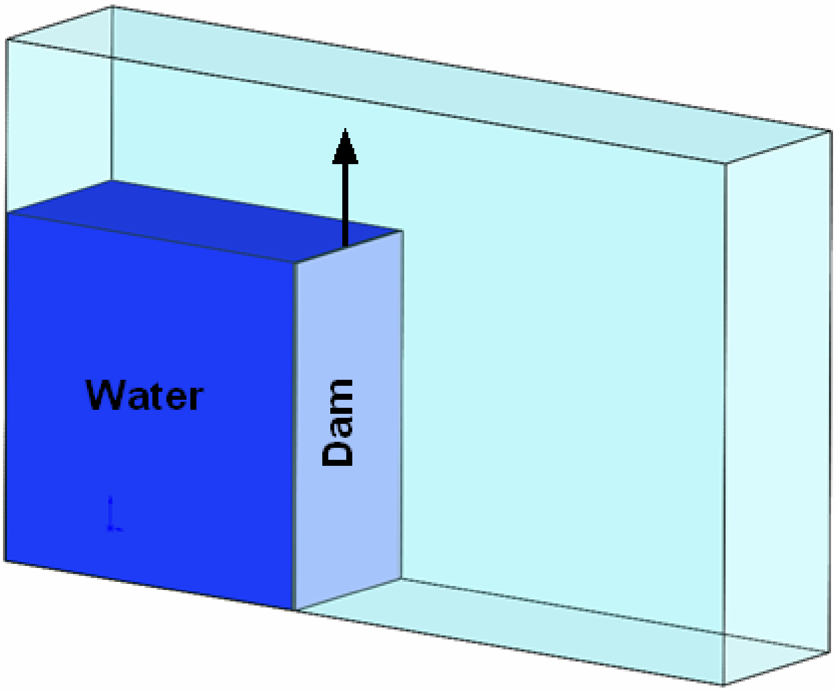
In *FlowVision* for simulating free surface the VOF method is implemented.

In order to simulate the movement of a liquid and take into account its free surface, you have to:
- In properties of the **Substance** specify **Aggregative state** = **Liquid**.
- In properties of the **Substance** you must specify the value of the **Density**.
- You have to create two **Phases**, then add into one **Phase** the **Substance** and enable there the computation of the equations of **Motion**, and the other **Phase** leave empty.
- Add both **Phases** into one **Model**, using the rule: the **Phase** with **Movement** should locate on the first place in the list, and the empty **Phase** should locate on the second place.
- Specify the appropriate boundary conditions for the variable **VOF** (volume of fluid, used in simulations with a free surface), and be sure that to specify somewhere in the computational domain the initial value **VOF=1**.
- Use the time step, which does not exceed 5×**Surface CFL**. When the shape of the free surface changes significantly, it is recommended to use the time step, which corresponds to 1×**Surface CFL**.

In order to simulate the movement of the two liquids or a liquid and gas based interface, you have to:
- Create two **Substances** and set their **Aggregate states** as **Liquid**/**Liquid** or **Liquid**/**Gas**.
- In properties of the **Substances** you must specify the value of their **Densities** or specify calculation of a **Density** according to the ideal gas law.
- Create two **Phases**, then add into each **Phase** an appropriate **Substance**, then enable the computation of the equations of **Motion**.
- Add both **Phases** into one **Model**, and the first on the list should be the **Phase**, which has the **Substance** with the bigger **Density**.
- Specify the appropriate boundary conditions for the variables **VOF** of each **Phase**, and be sure to specify somewhere in the computational domain the initial value **VOF=1**.
- Use the time step, which does not exceed 5×**Surface CFL**. When the shape of the free surface changes significantly, it is recommended to use the time step, which corresponds to 1×**Surface CFL**.

4.5.1 Broken dam

In this example, a collapse of a dam is simulated in two dimensions. A part of the computational domain is filled with water, which is confined by a wall (the dam). Initially, the dam is quickly removed and a wave begins to move.



Parameters of the problem setting

Dimensions:

Dimensions of the computational domain	a × b	= 5×3	[m × m]
Dimensions of the liquid's column	d × d	= 2×2	[m × m]

Fluid parameters:

Density	ρ	= 1000	[kg m^{-3}]
Viscosity	μ	= 0.001	[kg m^{-1}s^{-1}]

Geometry: `Wave.STL`

Project: `Wave`

4.5.1.1 Physical model

In the **Properties** window of the element **General settings** specify:

Gravity vector

X	**= 0**	[m s^{-2}]
Y	**=-9.8**	[m s^{-2}]
Z	**=0**	[m s^{-2}]

In the folder **Substances**:

- Create **Substance #0**.
- Specify the following properties of **Substance #0**:

Aggregative state	**= Liquid**	
Molar mass		
Value	**=0.018**	[kg mole^{-1}]
Density		
Value	**= 1000**	[kg m^{-3}]
Viscosity		
Value	**= 0.001**	[kg m^{-1}s^{-1}]
Specific heat		
Value	**= 4217**	[J kg^{-1} K^{-1}]

In the folder **Phases**:

- Create a continuous **Phase #0**.
- In **Phase #0** add **Substance #0** into the folder **Substances**.
- Specify properties of the folder **Phase #0 > Physical processes**:

Motion	**= Navier-Stokes model**
Turbulence	**= KES**

- Create a continuous **Phase #1**.

In the folder **Models**:

- Create **Model #0**.
- Add **Phase #0** and **Phase #1** into subfolder **Model #0 > Phases**.

Notes:

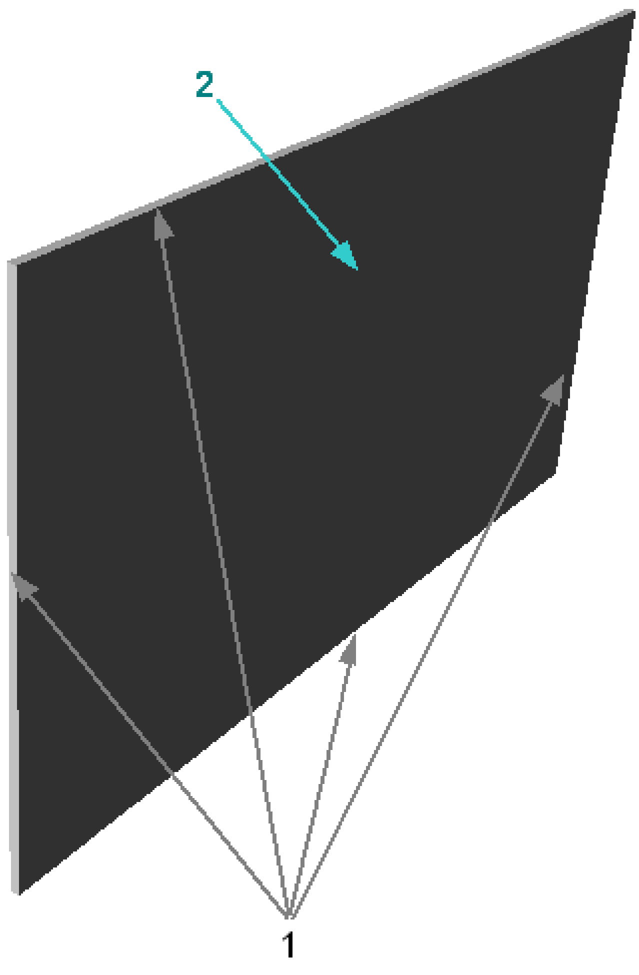
1. In this simulation, the flow in the second phase can be neglected, so the **Physical processes** are not set in **Phase #1**. And the **Phase**, in which physical processes are not specified, must always be the second in the folder **Model #N > Phases**.

2. After loading **Phase #0** and **Phase #1** into the **Model**, the **Phase interaction** element (of the **Continuum-vacuum** type) automatically appears and the **Phase transfer** element appears in **Physical processes** of **Phase #0**.

4.5.1.2 Boundary conditions

In the **Properties** window of the **SubRegion #0**, specify:

Model = Model #0



Specify the following boundary conditions:

Boundary 1

Type	**= Wall**
Variables	
Velocity(Phase #0)	**= Logarithm law**
TurbEnergy(Phase #0)	**= Value in cell near wall**
TurbDissipation(Phase #0)	**= Value in cell near wall**
VOF(Phase #0)	=**Symmetry**

Boundary 2

Type	**=Symmetry**
Variables	
Velocity(Phase #0)	**= Slip**
TurbEnergy(Phase #0)	**= Symmetry**
TurbDissipation(Phase #0)	**= Symmetry**
VOF(Phase #0)	**=Symmetry**

4.5.1.3 Specification of the water column

When simulating the of liquid's motion with taking into account the free surface, you always have to set the initial distribution of phases.

Specify the initial data in the **Properties** window of the element **Model #0 > Init. data > Init. data #0 > VOF(Phase #0)**:

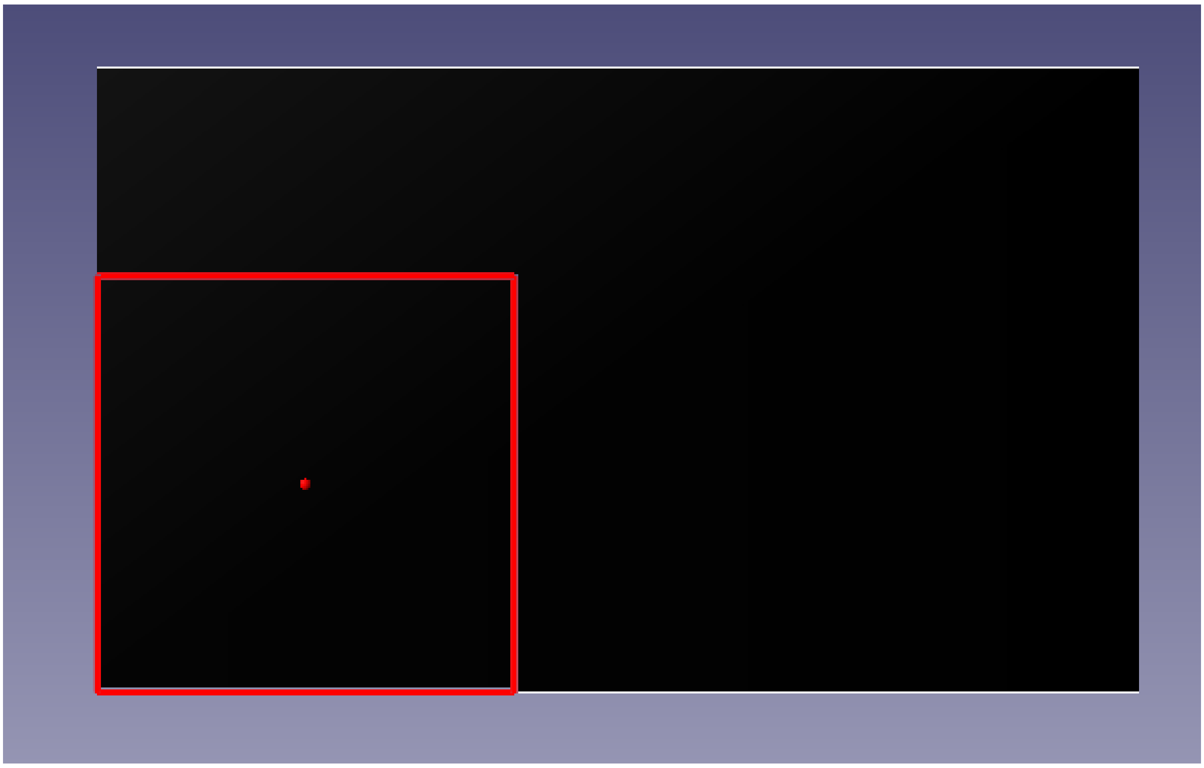
Value **= 1**

To define the area where the initial data are applied, do the following in the folder **Objects**:

- Create **Box #0**.
- In properties of **Box #0** specify:

Location		
Reference point		
X	**= 1**	[m]
Y	**= 1**	[m]
Z	**= 0.025**	[m]
Size		
X	**=2**	[m]
Y	**=2**	[m]
Z	**= 0.05**	[m]

In the **View** window an image of **Box #0** will appear:

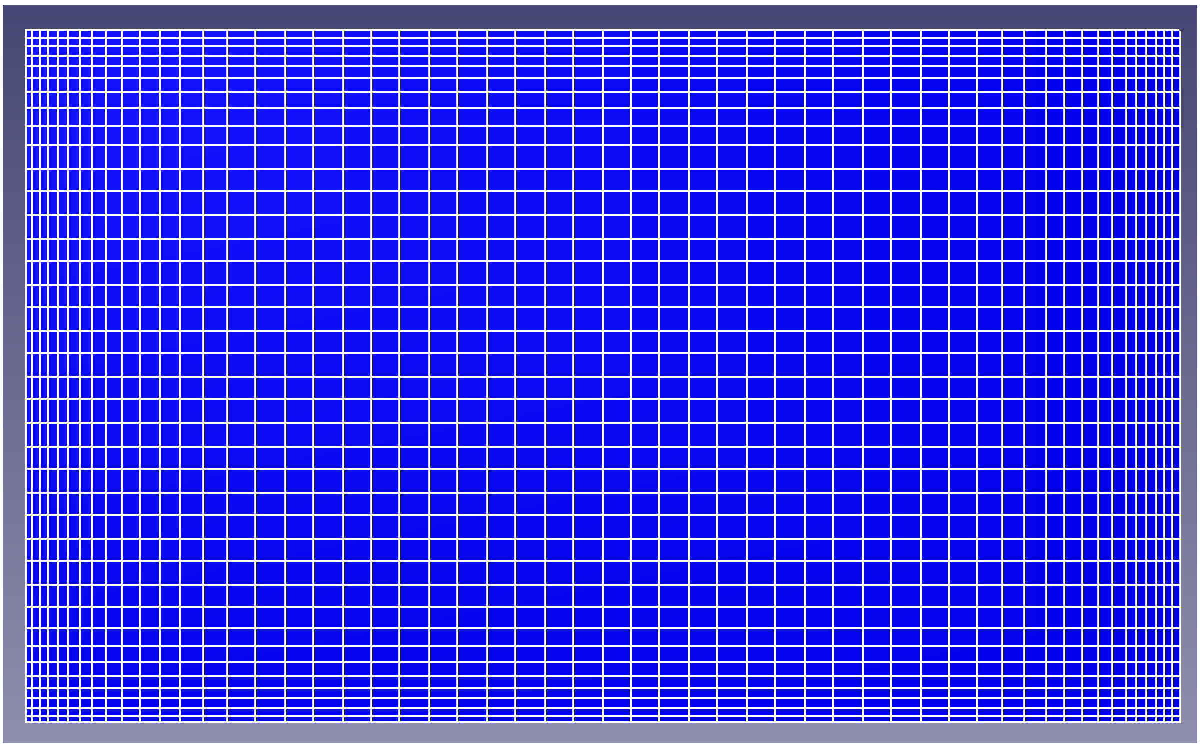


To define the matching between the geometrical **Object** and the **Initial data**, specify in properties of the element **SubRegion #0 > Initial conditions > Initial. conditions #0**:

Object	**= Box #0**
Init. data	**= Init. data #0**
Method	**= Replace in cropped volume**

Thus, in the volume of the box, a variable is set as **VOF=1**, which corresponds to the presence (at the initial time moment) a water column with dimensions 2×2 [m].

4.5.1.4 Initial grid



In the **Properties** window of the **Initial grid**, click the button to open the **Initial grid editor**.

Specify in the **Initial grid editor**:

for axis OX

Grid parameters

h_max = 0.125 [m]

h_min = 0.03 [m]

Insert reference lines with coordinates:

x1 = 1 [m]

x2 = 4 [m]

Specify **Reference line parameters** for the reference line with coordinate **x=0**: [m]

h = 0.03 [m]

Specify **Reference line parameters** for the reference line with coordinate **x=1**:

h = 0.125 [m]

Specify **Reference line parameters** for the reference line with coordinate **x=4**:

h = 0.125 [m]

Specify **Reference line parameters** for the reference line with coordinate **x=5**:

h = 0.03 [m]

for axis OY (click the button Y)

Grid parameters

h_max = 0.1 [m]

h_min = 0.03 [m]

Insert reference lines with coordinates:

y1 = 0.5 [m]

y2 = 2.5 [m]

Specify **Reference line parameters** for the reference line with coordinate **y=0**:

h	**= 0.03**	[m]

Specify **Reference line parameters** for the reference line with coordinate **y=0.5**:

h	**= 0.1**	[m]

Specify **Reference line parameters** for the reference line with coordinate **y=2.5**:

h	**= 0.1**	[m]

Specify **Reference line parameters** for the reference line with coordinate **y=3**:

h	**= 0.03**	[m]

Click **OK** to close the **Initial grid editor** with saving the entered data.

In properties of the **Initial grid** specify:

Grid structure = 2D

Plane = XY

In the **Properties** window of the **Initial grid** click **Apply**.

4.5.1.5 Parameters of calculation

Specify in the **Solver** tab in properties of the **Time step** element:

Method	**= Via CFL number**	
Convective CFL	**= 1**	
Surface CFL	**= 1**	
Max step	**= 0.01**	[s]

4.5.1.6 Visualization

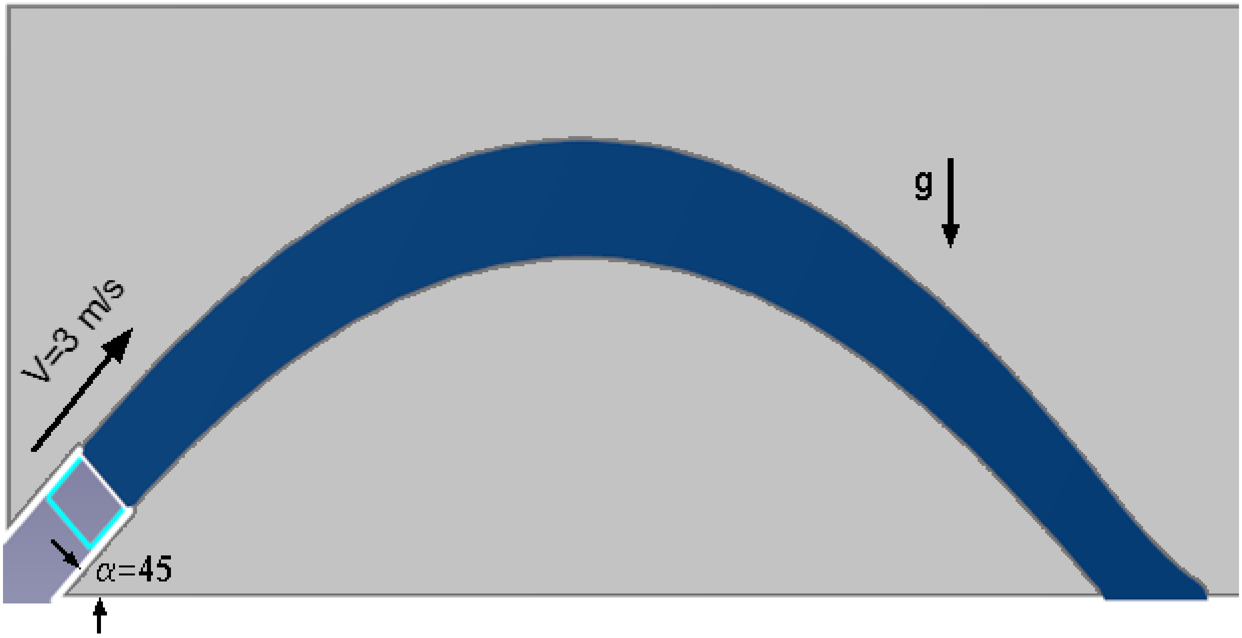
To view the dynamics of the solution during the computation, specify visualization of liquid's distribution in the plane of symmetry before the start of computation.

4.5.1.6.1 Distribution of the liquid

- Create a **VOF** layer on the **Computational space**.

4.5.2 Free jet

In this example, consider the motion of a free two-dimensional jet in the gravity field.



Parameters of the problem setting

Free fall acceleration	g	= 9.8	[m s^{-2}]
Fluid parameters:			
Density	ρ	= 1000	[kg m^{-3}]
Viscosity	μ	= 0.001	[kg $m^{-1}s^{-1}$]
Inlet parameters:			
Velocity of the flow	V	= 3	[m s^{-1}]
Initial angle between the flow and the horizon	α	= 45	
Geometry	`FreeJet.STL`		
Project:	`FreeJet`		

4.5.2.1 Physical model

In the **Properties** window of the element **General settings** specify:

Gravity vector

X	**= 0**	[m s^{-2}]
Y	**=-9.8**	[m s^{-2}]
Z	**=0**	[m s^{-2}]

In the folder **Substances**:

- Create **Substance #0.**
- Specify the following properties of **Substance #0**:

Aggregative state **= Liquid**

Molar mass

Value =0.018 [kg $mole^{-1}$]

Density

Value = 1000 [kg m^{-3}]

Viscosity

Value = 0.001 [kg $m^{-1}s^{-1}$]

Specific heat

Value = 4217 [J kg^{-1} K^{-1}]

In the folder **Phases**:

- Create a continuous **Phase #0**.
- In **Phase #0** add **Substance #0** into the folder **Substances**.
- Specify properties of the folder **Phase #0 > Physical processes**:

Motion = **Navier-Stokes model**

- Create a continuous **Phase #1**.

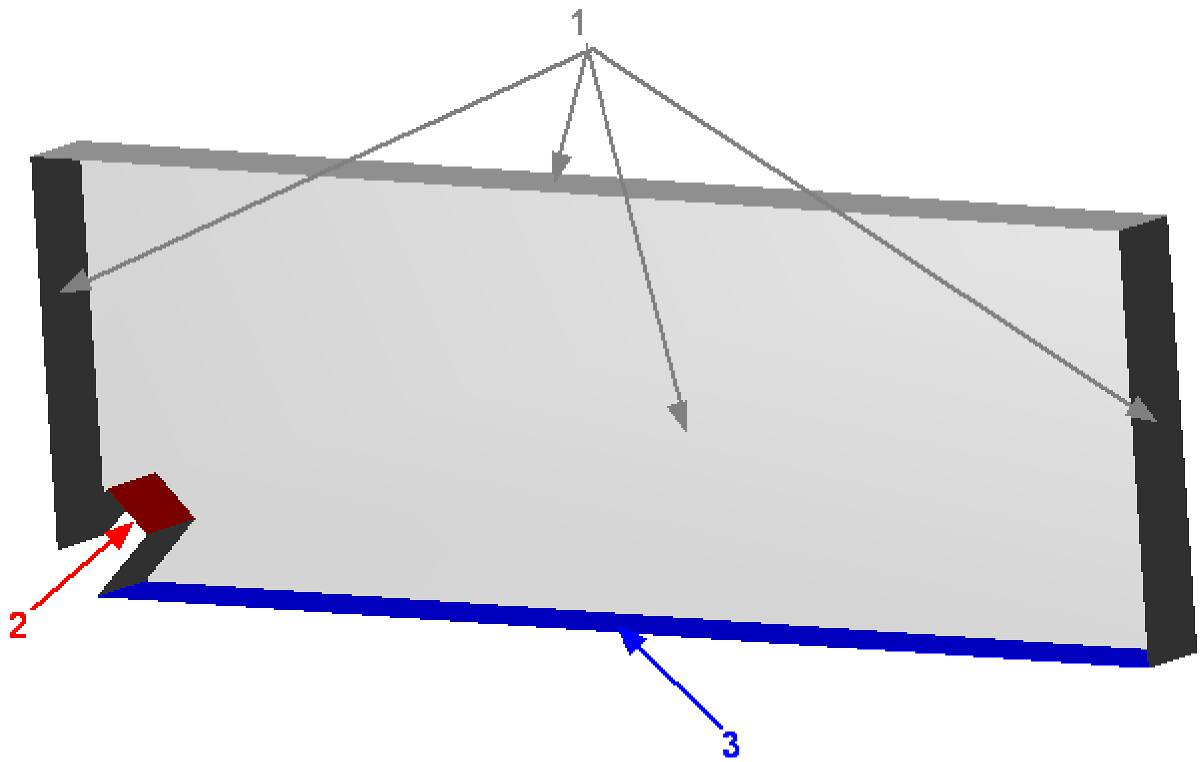
In the folder **Models**:

- Create **Model #0**.
- Add **Phase #0** and **Phase #1** into subfolder **Model #0 > Phases**.

4.5.2.2 Boundary conditions

In the **Properties** window of the **SubRegion #0**, specify:

Model = Model #0



Specify the following boundary conditions:

Boundary 1

Type = **Symmetry**

Variables

Velocity (Phase #0) = **Slip**

VOF (Phase #0) =**Symmetry**

Boundary 2

Type	**= Inlet/Outlet**	
Variables		
Velocity (Phase #0)	**= Normal mass velocity**	
Mass velocity	**= 3000**	[kg m^{-2} s^{-1}]
VOF (Phase #0)	**= Value**	
Value	**= 1**	

Boundary 3

Type	**= Free Outlet**	
Variables		
Velocity (Phase #0)	**= Pressure**	
Value	**= 0**	[Pa]
VOF (Phase #0)	**= Zero gradient**	

4.5.2.3 Initial conditions

In simulations with a free surface it is necessary that some volume of the liquid be present in the region at the initial time moment.

Specify the initial volume of liquid in the area where the flow goes into the subregion without the liquid:

- In properties of the element **Model #0 > Init. data #0** specify:

Velocity			
	X	**= 2.12**	[m s^{-1}]
	Y	**= 2.12**	[m s^{-1}]
	Z	**= 0**	[m s^{-1}]
VOF			
	Value	**= 1**	

- In the folder **Objects**, create **Box #0**.
- In properties of **Box #0** specify:

Location				
	Reference point			
		X	**= 0.1**	[m]
		Y	**= 0.1**	[m]
		Z	**= 0.05**	[m]
	Axis X			
		X	**= 1**	
		Y	**= 1**	
		Z	**= 0**	
Size				
	X		**= 0.1**	[m]
	Y		**= 0.07**	[m]
	Z		**= 0.1**	[m]

- In properties of the element **SubRegion #0 > Initial conditions > Init. condition #0** specify:

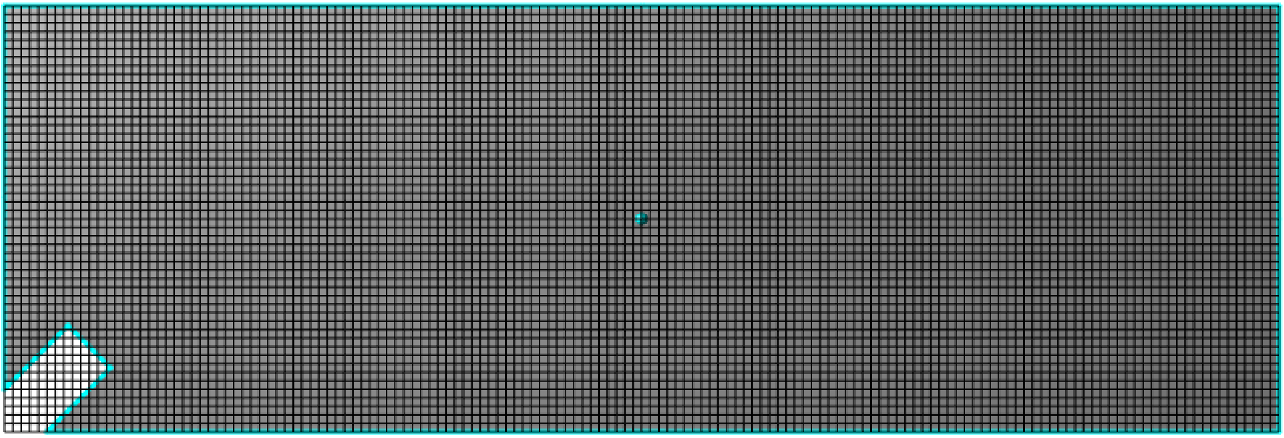
Object	**= Box #0**
Init. data	**= Init. data #0**

Method = **Replace in cropped volume**

Note:

After clicking the **Apply** button in the **Properties** window of **Box #0**, the vectors of the coordinate system of the box will be automatically redirected and orthonormalized.

4.5.2.4 Initial grid



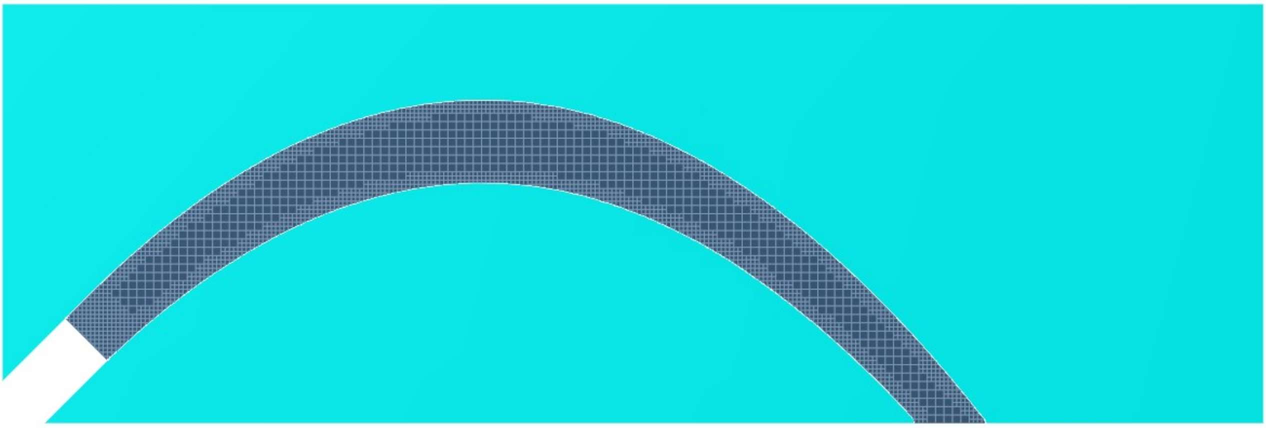
Specify in the **Properties** window of the **Initial grid**:

Grid structure = 2D
Plane = XY
nX = 150
nY = 50

In the **Properties** window of the **Initial grid** click **Apply**.

4.5.2.5 Adaptation on the inter-phase surface

For better accuracy of simulating multiphase flows, it is recommended to use more refined computational grid on the inter-phase surface. You can do this by applying adaptation of the computational grid that will become active depending on value of the variable **VOF** (when **0 < VOF < 1** in a cell, the free surface passes through this cell).



As in this exercise the inter-phase surface moves over the computational domain, use of only an **Adaptation by condition** will cause refining the grid even in areas where it is not necessary anymore (due to the inter-phase surface changed its location) as shown on the illustration below:

The **Computational grid section** layer when only **Adaptation by condition** is used (no merging is done in the areas, from which the free surface has get away). The grid contains 920 computational cells.

To prevent waste of the computational grid, you should to set adaptation on the free surface; this adaptation will move together with the jet and will refine the grid near the inter-phase surface only:

The **Computational grid section** layer when adaptation on the moving inter-phase surface is used. The grid contains 590 computational cells.

The adaptation on the moving inter-phase surface consists of combination of the three adaptation listed below:

1. A simple **Adaptation** inside the geometry object, in which **Initial conditions** were set (this object is **Box #0**). In the project tree, in the folder **Computational grid > Adaptation** create **Adaptation #0** within the object **Box #0**. In properties of **Adaptation #0** specify:
Enabled = Yes
Max level N = 1
Split/Merge = Split
Layers > Layers for Level N = 3

Grid adaptation within the object with **Initial conditions** is required only until obtaining a correct solution on the first computational step. After computing the first step, you can disable **Adaptation #0.**

The grid adaptation settings are applied before placing the free surface, so surface of the jet will not be resolved yet by the grid on the first computational step. And when the adaptation level changes on the next computational step, the inter-phase surface might move away. So the rule follows from this: the adaptation level near the inter-phase surface should not change during the whole computation.

2. **Adaptation by condition**, which will be working depending on value of the variable **VOF.** In the project tree, in the folder **Computational grid > Adaptation by condition** create **Adaptation by condition #0** and in its properties specify:
Enabled = Yes
Max level N = 1
Layers > Layers for Level N = 3

Conditions > Variable > Category = Variables of phase "Phase #0"
Conditions > Variable > Variable = VOF
Conditions > Range > From = 0.0001
Conditions > Range > To = 0.9999

Specifying **Range** of the variable **VOF** from **0.0001** to **0.9999** provides applying **Adaptation by condition #0** just on the inter-phase surface (because **VOF (Phase #0) = 1** within the jet, **VOF (Phase #0) = 0** outside the jet, and **0 < VOF (Phase #0) < 1** on the inter-phase surface). You can set other limits of the range within values **0 < VOF (Phase #0) < 1** but more closer are the limits to values **0** and **1**, the more accurate resolution of the free surface by the grid will be.

From the context menu of the element **Adaptation by condition > Adaptation by condition #0 > Object** select the command **Add/Remove Objects** and in the dialog box, which opens, add the **Computational space** object into the list of the selected objects and then click **OK**.

3. A simple **Adaptation** that merges cell. This adaptation will cancel the grid refining that was made by **Adaptation by condition #0**, it restores the initial size of cells in the areas, from which the jet get away. In the project tree, in the folder **Computational grid > Adaptation** create **Adaptation #1** within the object **Computational space** and in its properties specify:

Enabled = Yes
Max level N = 0
Split/Merge = Merge

4.5.2.6 Parameters of calculation

Specify in the **Solver** tab in properties of the **Time step** element:

Method	**= Via CFL number**	
Convective CFL	**= 1**	
Surface CFL	**= 1**	
Max step	**= 0.01**	[s]

4.5.2.7 Visualization

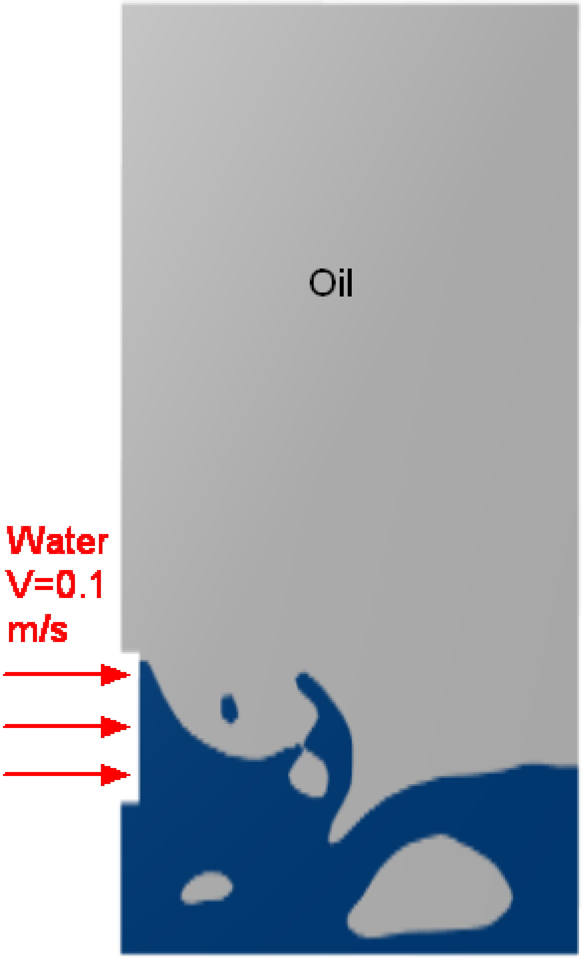
To view the dynamics of the solution during the computation, specify visualization of: liquid's distribution before the start of computation.

4.5.2.7.1 Distribution of the liquid

- Create a **VOF** layer on the **Computational space**.

4.5.3 Displacement of oil by water

In this example, consider the displacement oil by water from a tank. At the initial moment of the time, the tank is completely filled with oil. From the hole in the side wall of the tank the water comes into and gradually displace the oil from the tank.



Parameters of the problem setting

Parameters of water:

Density	ρ	= 1000	[kg m^{-3}]
Viscosity	μ	= 0.001	[kg m^{-1}s^{-1}]
Surface tension		= 0.073	[N m^{-1}]

Parameters of oil:

Density	ρ	= 500	[kg m^{-3}]
Viscosity	μ	= 0.01	[kg m^{-1}s^{-1}]
Surface tension		= 0.0647	[N m^{-1}]

Inlet parameters:

Velocity of water, which is being fed	V	= 0.1	[m s^{-1}]

Geometry **`TwoFluids.wrl`**

Project: **`TwoFluids`**

4.5.3.1 Physical model

In the **Properties** window of the element **General settings** specify:

- Specify the following parameters:

Gravity vector			
	X	**= 0**	[m s^{-2}]
	Y	**=-9.8**	[m s^{-2}]
	Z	**=0**	[m s^{-2}]
g-Point			
	X	**= 0**	[m]
	Y	**= 0.25**	[m]
	Z	**= 0**	[m]
g-Density		**= 500**	[kg m^{-3}]

In the folder **Substances**:

- Create **Substance #0**.
- Specify the following properties of **Substance #0**:

Aggregative state	**= Liquid**	
Molar mass		
Value	**=0.018**	[kg $mole^{-1}$]
Density		
Value	**= 1000**	[kg m^{-3}]
Viscosity		
Value	**= 0.001**	[kg $m^{-1}s^{-1}$]
Specific heat		
Value	**= 4217**	[J kg^{-1} K^{-1}]
Surface tension		
Value	**= 0.073**	[N m^{-1}]

- Create **Substance #1**.
- Specify the following properties of **Substance #1**:

Aggregative state	**= Liquid**	
Molar mass		
Value	**=0.009**	[kg $mole^{-1}$]
Density		
Value	**= 500**	[kg m^{-3}]
Viscosity		
Value	**= 0.01**	[kg $m^{-1}s^{-1}$]
Specific heat		
Value	**=2688**	[J kg^{-1} K^{-1}]
Surface tension		
Value	**=0.0647**	[N m^{-1}]

In the folder **Phases**:

- Create a continuous **Phase #0**.

- In **Phase #0** add **Substance #0** into the folder **Substances**.
- Specify properties of the folder **Phase #0 > Physical processes**:

  Motion **= Navier-Stokes model**

- Create a continuous **Phase #1**.
- In **Phase #1** add **Substance #1** into the folder **Substances**.
- Specify properties of the folder **Phase #1 > Physical processes**:

  Motion **= Navier-Stokes model**

In the folder **Models**:

- Create **Model #0.**
- Add **Phase #0** and **Phase #1** into subfolder **Model #0 > Phases**.

Notes:

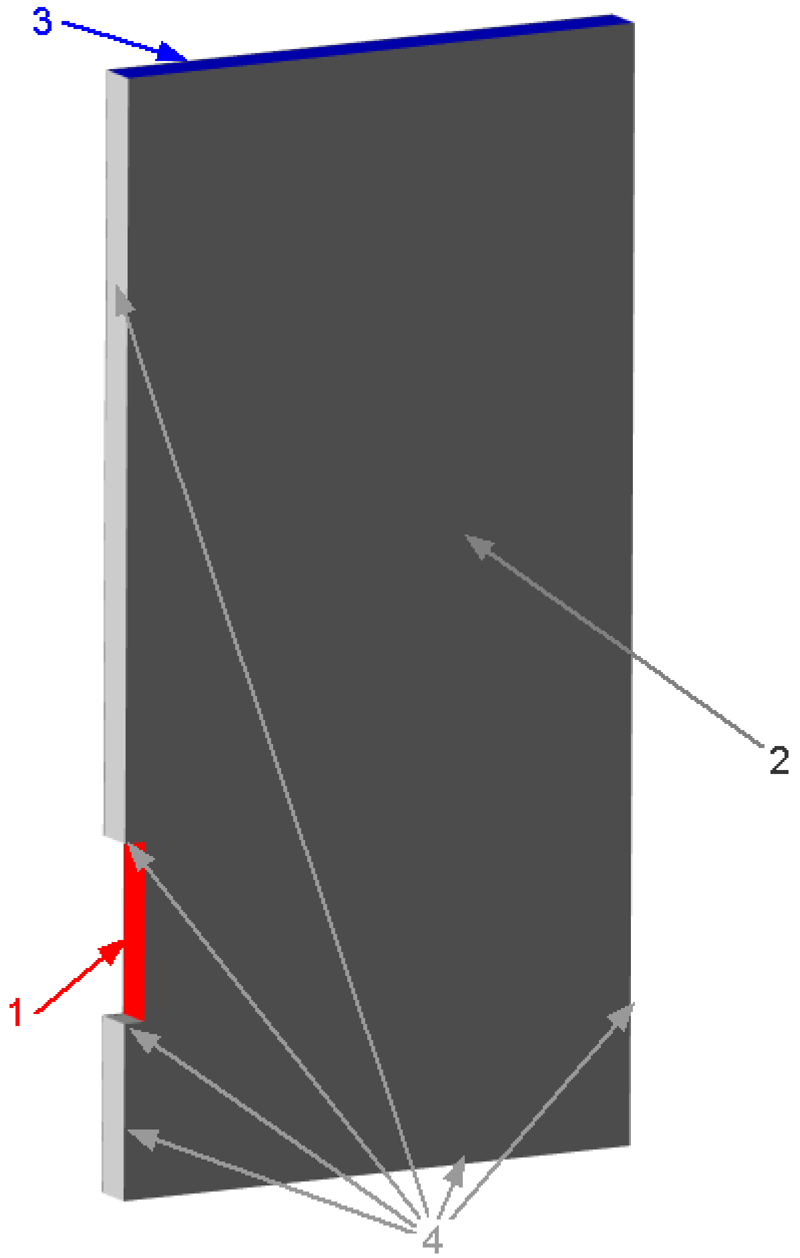
1. In this simulation, **Phase #0** should be placed on the first position in the **Model**, because its density is bigger then density of **Phase #1**.

2. After loading **Phase #0** and **Phase #1** into the **Model**, the **Phase interaction** element (of the **Continuum-continuum** type) automatically appears and the **Phase transfer** element appears in **Physical processes** of **Phase #1**.

4.5.3.2 Boundary conditions

In the **Properties** window of the **SubRegion #0**, specify:

Model = Model #0



Specify the following boundary conditions:

Boundary 1

Type	**= Inlet/Outlet**	
Variables		
Velocity (Phase #0)	**= Normal mass velocity**	
Mass velocity	**= 100**	[kg $m^{-2}s^{-1}$]
VOF (Phase #0)	**= Value**	
Value	**= 1**	
Velocity (Phase #1)	**= Normal mass velocity**	
Mass velocity	**= 0**	[kg $m^{-2}s^{-1}$]
VOF (Phase #1)	**= Value**	

Value **= 0**

Boundary 2

Type **= Symmetry**

Variables

Velocity(Phase #0) **= Slip**

VOF (Phase #0) **= Symmetry**

Velocity(Phase #1) **= Slip**

VOF (Phase #1) **= Symmetry**

Boundary 3

Type **= Free Outlet**

Variables

Velocity(Phase #0) **= Pressure**

Value **= 0** [Pa]

VOF (Phase #0) **= Zero gradient**

Velocity(Phase #1) **= Pressure**

Value **= 0** [Pa]

VOF (Phase #1) **= Zero gradient**

Boundary 4

Type **= Wall**

Variables

Velocity(Phase #0) **= No slip**

VOF (Phase #0) **= Symmetry**

Velocity(Phase #1) **= No slip**

VOF (Phase #1) **= Symmetry**

4.5.3.3 Initial conditions defining volumes of liquids

In **Model #0**:

- In **Init. data #0** specify:

VOF(Phase #1)

Value **= 0**

- Create **Init. data #1**.
- In **Init. data #1** specify:

Velocity(Phase #0)

Value

X **= 0.1** [m s^{-1}]

VOF(Phase #0)

Value **= 1**

> Before creating **Init. condition #1** you have to create **Box #0**, which is a bit smaller than the inlet.
> In properties of **Init. condition #1**, which will be created on **Box #0**, **Method = Replace in full volume** will be specified, so the water will initially fill the whole inlet due to cells that even partially contact **Box #0.**

In the folder **Objects**:

- Create **Box #0**.

- In the **Properties** window of **Box #0** specify:

Location			
Reference point			
X		**= 0**	[m]
Y		**= 0.06**	[m]
Z		**= 0.005**	[m]
Size			
X		**= 0.04**	[m]
Y		**= 0.035**	[m]
Z		**= 0.01**	[m]

In **SubRegion #0**:

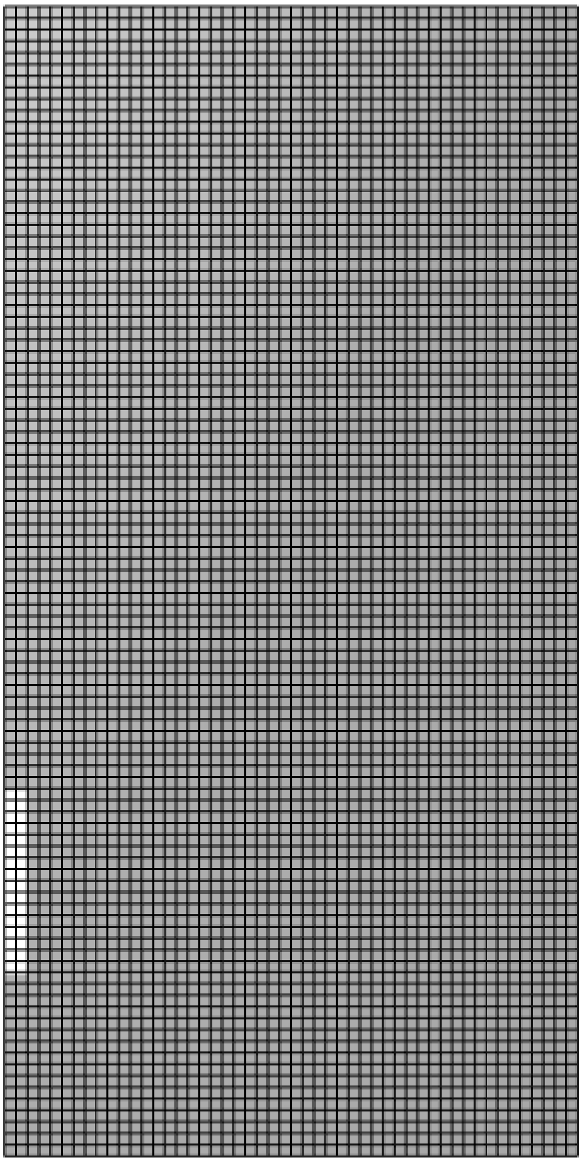
- Make sure that in properties of the element **Initial conditions > Init. condition #0** the following settings are specified:

Object	**= Computational space**
Init. data	**= Init. data #0**

- Create the element **Initial conditions > Init. condition #1**.
- In properties of **Init. condition #1** specify:

Object	**= Box #0**
Init. data	**= Init. data #1**
Method	**= Replace in full volume**

4.5.3.4 Initial grid



Specify in the **Properties** window of the **Initial grid**:

Grid structure = 2D
Plane = XY
nX = 50
nY = 100

In the **Properties** window of the **Initial grid** click **Apply**.

4.5.3.5 Parameters of calculation

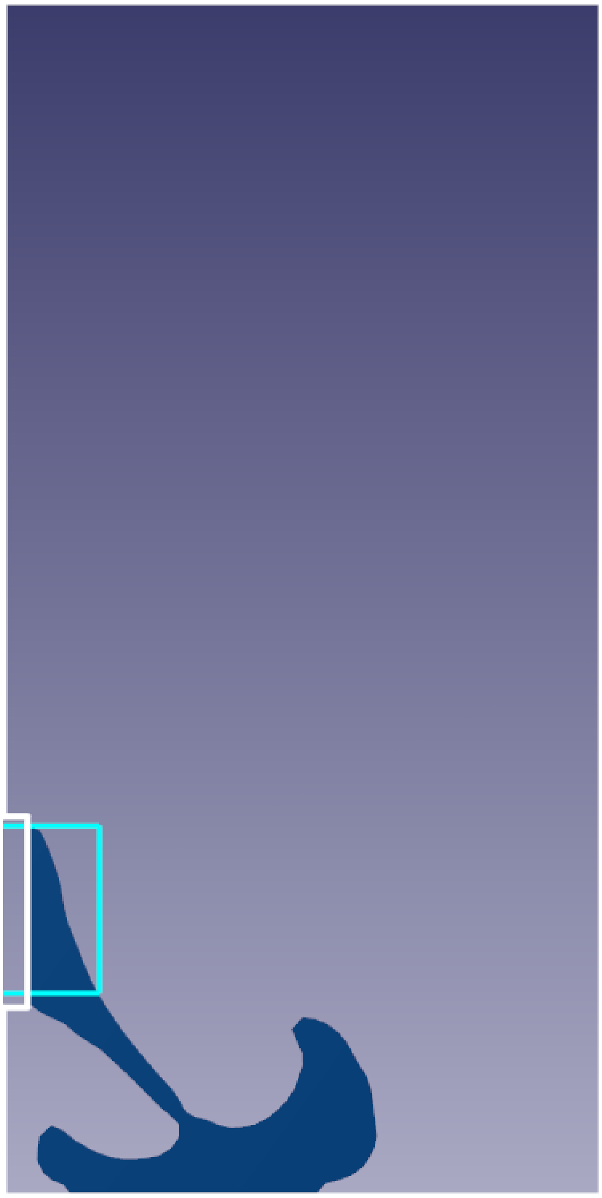
Specify in the **Solver** tab in properties of the **Time step** element:

Method	**= Via CFL number**	
Surface CFL	**= 1**	
Max step	**= 1**	[s]

4.5.3.6 Visualization

To view the dynamics of the solution during the computation, specify visualization of the liquid's surface before the start of computation.

4.5.3.6.1 Water distribution



- Create a **VOF** layer on the **Computational space**.

4.6 Dispersed media

The implemented model of dispersed medium allows solving the following problems:

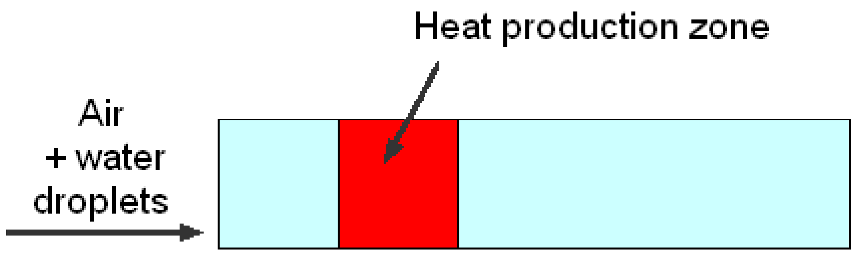
- flow of liquid/gas in porous carcass
- motion of solid particles in liquid/gas
- motion of droplets in liquid/gas
- motion of bubbles in liquid

We provide the following exercises related to dispersed media:

- Droplet evaporation in air
- Coal combustion

4.6.1 Droplet evaporation in air

Consider the evaporation of droplets in the air because of being heated:



Parameters of the problem setting

Inflow parameters:

Mass flow rate of air:	V	= 0.5522	[kg m^{-2} s^{-1}]
Air temperature:	T_{air}	= 10	[K]
Water droplets' temperature	T_{drops}	= 0	[K]
Diameter of the water droplets	d	= 0.0001	[m]
Fraction of water droplets:		= 0.001	

Parameters of the heat source

Power	P	= 2e6	[W m^{-3}]

Geometry: `Drops.wrl`

Project: `Drops`

4.6.1.1 Physical model

In the folder **Substances**:

- Create **Substance #0**.
- Load **Substance #0** from the **Standard** substance database:
  - From the context menu of **Substance #0** select **Load from SD > Standard.**
  - In the new window **Load from database**, select:

    Substances **= Air**

    Phases **= Gas (equilibrium)**
- Create **Substance #0.**
- Load the properties of **Substance #0** from the **Substance Database**:

  Substance **= Water**

Phases **=Gas (equilibrium)**

- Create **Substance #0**.
- Load the properties of **Substance #0** from the **Substance Database**:

Substances **= Water**

Phases **= Liquid**

In the folder **Phases**:
- Create a continuous **Phase #0.**
- Add the following substances in the folder **Substances**:

Water_Gas (equilibrium)
Air_Gas (equilibrium)

- Specify properties of the folder **Phase #0 > Physical processes**:

Heat transfer **= Heat transfer via h**

Motion **= Navier-Stokes model**

Mass transfer **= Chemistry**

- Create a dispersed **Phase #1** of the **Particles** type (open the context menu of the folder **Phases** and select there the **Create particles** command)
- Add **Water_Liquid** to folder **Substances** of **Phase #1.**
- Specify properties of the folder **Phase #1 > Physical processes**:

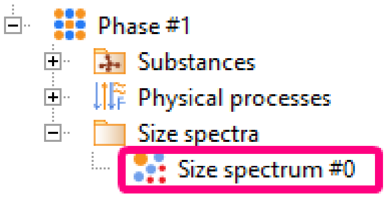
Heat transfer **= Convection & conduction**

Phase transfer **= Convection & diffusion**

Motion **= Motion**

Mass transfer **= Mass transfer**

- From the context menu of the folder **Phase #1 > Size spectra** select the **Create** command. The element **Phase #1 > Size spectra > Size spectrum #0** will appear in the project tree:



- As **Number of size groups = 1** is specified in properties of **Phase #1** (this is the default value), **Size spectrum #0** will contain only one size group of particles and the **Size groups > [0] > Volume fraction in the Phase** parameter will be read-only and have the value **1.** Specify diameter of particles in the size group **[0]** of **Size spectrum #0**:

Size groups > [0] > Diam. particles= 0.0001 [m]

This prepared **Size spectrum #0** will be used in **Initial data** and on **Boundary conditions** for inlet.

In the folder **Models**:
- Create **Model #0.**
- Add **Phase #0** and **Phase #1** into subfolder **Model #0 > Phases.**
- In the folder **Phase interaction** in properties of the element **Continuum-particles** do:
  - Click **Add** () near the element **Substance pair.**
  - Specify:

Substance pair

[0]

Phase0 = **Water_Gas (equilibrium)**

Phase1 = **Water_Liquid**

Cd **= Model1**

Nu	**= Model1**
Evaporation model	**= Model1**
Sh	**= Model1**

- For the element **Init. data > Init. data #0** specify:

Temperature (Phase #0)

Value	**= 0**	[K]

Velocity (Phase #0)

Value

X	**= 1**	[m s^{-1}]
Y	**= 0**	[m s^{-1}]
Z	**= 0**	[m s^{-1}]

Mass frac._Water_Gas (equilibrium) (Phase #0)

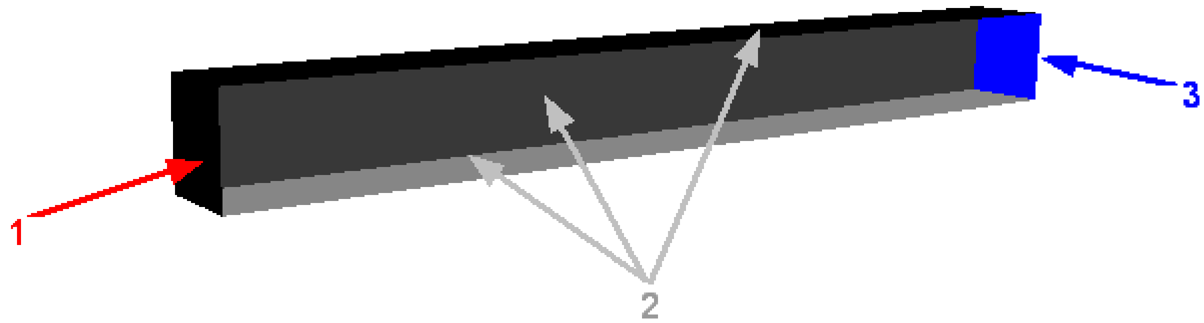
Value	**= 0**

Mass frac._Air_Gas (equilibrium) (Phase #0)

Value	**= 1**

In properties of **SubRegion #0**, specify: **Model = Model #0**.

4.6.1.2 Boundary conditions



Specify the following boundary conditions:

Boundary 1

Type	**= Inlet/Outlet**	
Variables		
Temperature(Phase #0)	**= Temperature**	
Value	**= 10**	[K]
Velocity(Phase #0)	**= Normal mass velocity**	
Mass velocity	**= 0.5522**	[kg m^{-2} s^{-1}]
Mass frac. [Water_Gas (equilibrium)] (Phase #0)	**= Value at the inlet**	
Value	**= 0**	
Mass frac. [Air_Gas (equilibrium)] (Phase #0)	**= Value at the inlet**	
Value	**= 1**	
Phase volume(Phase #1)	**= Value**	
Value	**= 0.001**	

Temperature (disp.)(Phase #1)	**= Value**	
Value	**= 0**	[K]
Velocity (disp.)(Phase #1)	**= Particles velocity**	
Particles velocity		
X	**= 1**	[m s^{-1}]
Y	**= 0**	[m s^{-1}]
Z	**= 0**	[m s^{-1}]
Diameter(Phase #1)	**= Size spectrum**	
Size spectrum	**= Size spectrum #0**	

Boundary 2

Type	**= Symmetry**
Variables	
Temperature(Phase #0)	**= Symmetry**
Velocity(Phase #0)	**= Slip**
Mass frac. [Water_Gas (equilibrium)] (Phase #0)	**= Symmetry**
Mass frac. [Air_Gas (equilibrium)] (Phase #0)	**= Symmetry**
Phase volume(Phase #1)	**= Symmetry**
Temperature (disp.)(Phase #1)	**= Symmetry**
Velocity (disp.)(Phase #1)	**= Contact with wall**
Diameter (Phase #1)	**= Symmetry**

Boundary 3

Type	**= Free Outlet**	
Variables		
Temperature(Phase #0)	**= Zero gradient**	
Velocity(Phase #0)	**= Pressure**	
Value	**= 0**	[Pa]
Mass frac. [Water_Gas (equilibrium)] (Phase #0)	**= Zero gradient**	
Mass frac. [Air_Gas (equilibrium)] (Phase #0)	**= Zero gradient**	
Phase volume(Phase #1)	**= Permeable surface**	
Temperature (disp.)(Phase #1)	**= Permeable surface**	
Velocity (disp.)(Phase #1)	**= Permeable surface**	
Diameter (Phase #1)	**= Permeable surface**	

4.6.1.3 Modifiers

In order to set the **Volume heat source** modifier, do the following:

- From the context menu of the folder **Objects**, select **Create**.
- In the **Create new object** window, which appears, select **Box**.
- In the **Properties** window of **Box #0**, specify:

Location		
Reference point		
X	**= 0.15**	[m]

Y	**= 0.05**	[m]
Z	**= 0.05**	[m]
Size		
X	**= 0.1**	[m]
Y	**= 0.1**	[m]
Z	**= 0.1**	[m]

- In the context menu of the **Modifiers** folder, select **Create**.
- In the **Create new modifier** window, specify:

Modifier type	**= Volume heat source**
Objects	**= Box #0**

- In properties of the **Volume heat source #0** modifier, which appears, specify:

Activation		
Type	**= Permanent**	
Volume heat source	**= 2e6**	[W m^{-3}]

4.6.1.4 Initial grid

Specify in the **Properties** window of the **Initial grid**:
Grid structure = 1D
Direction = X
nX = 104

In the **Properties** window of the **Initial grid** click **Apply**.

4.6.1.5 Parameters of calculation

Specify in the **Solver** tab:

- In properties of the **Time step** element:

Method	**= Via CFL number**
Convective CFL	**= 1**
Diffusive CFL	**= 1e+20**

- In properties of the **Advanced settings** element specify:

Numerical method	
Type of scheme	**= Implicit**
Pressure gradient	**= Simple**
Multiphase D	
Cloud boundary	**= 0.0001**

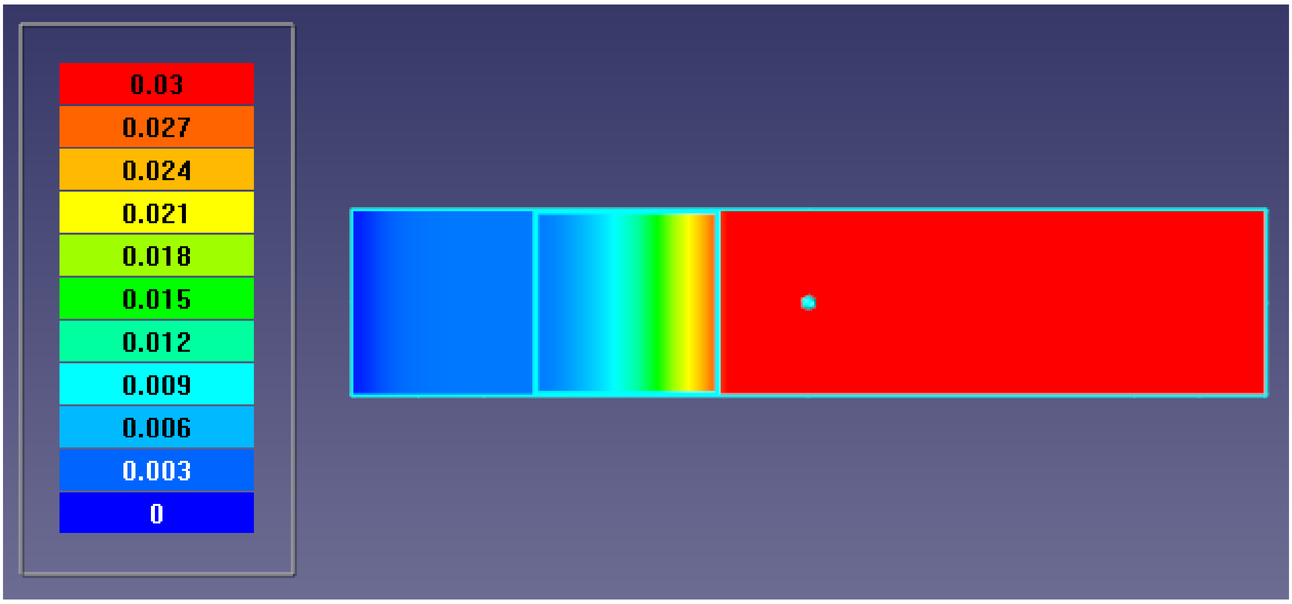
4.6.1.6 Visualization

Specify visualizations of:

1. Moisture vapor distribution in the plane of the flow
2. Temperature distribution in the plane of the flow

4.6.1.6.1 Moisture vapor distribution

Visualization at the step 450:



- In the **Properties** window of **Plane #0** specify:

Object

Normal

X **= 0**

Y **= 0**

Z **= 1**

(to direct a **Plane**'s normal along the axis **Z**, you can also click the **Operations >** button in the **Properties** window of the **Plane**)

- Create a layer **Color contours** on **Plane #0**.
- In the **Properties** window of the layer specify:

Variable

Category = **Variables of phase "Phase #0"**

Variable **= Mass. frac [Water_Gas (equilibrium)]**

Value range

Mode **= Manual**

Max **= 0.03**

Min **= 0**

Palette

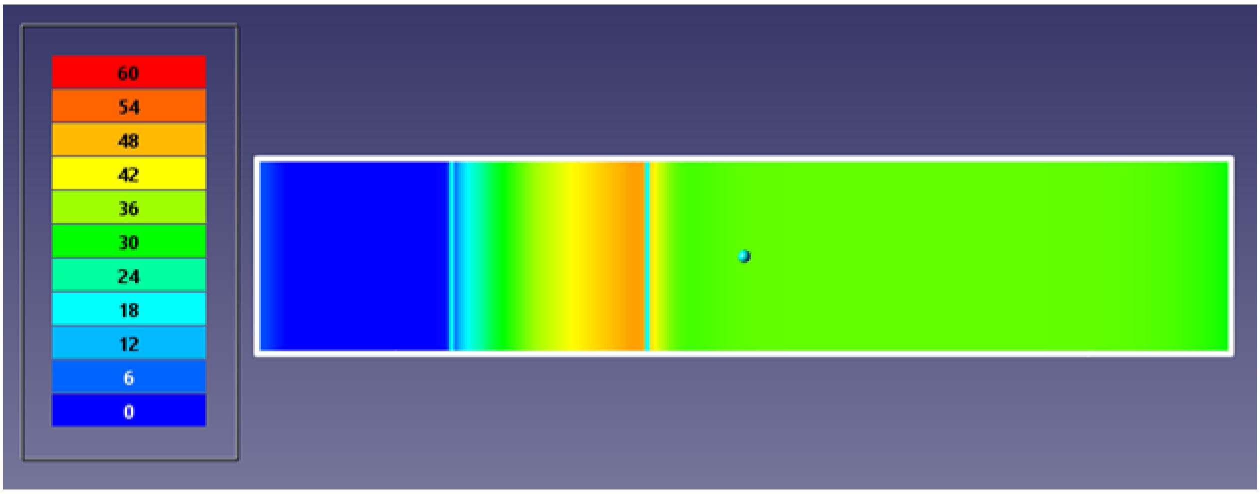
Appearance

Enabled **= Yes**

Style **= Style 1**

4.6.1.6.2 Temperature distribution

Visualization at the step 450:



- Create a layer **Color contours** on **Plane #0**.
- In the **Properties** window of the layer specify:

Variable			
	Variable		**= Temperature**
Value range			
	Mode		**= Manual**
	Max		**= 60**
	Min		**= 0**
Palette			
	Appearance		
		Enabled	**= Yes**
		Style	**= Style 1**

4.6.2 Coal combustion

In this example we simulate coal combustion in an 1D problem setting.

Mixture of gas and coal particles is being blown in a channel with high initial temperature 2073 [K] where the coal begins to burn due to the high temperature.

Parameters of the problem setting

Inflow parameters:

Mass velocity of the gas	$u1$	= 31.73	[kg/(m^2 s)]
Temperature of the gas	T	= 573	[K]
Mass fraction of oxygen	$m[O2]$	= 0.23333	
Mass fraction of coal in the mixture	n	= 0.0003	
Mass velocity of coal particles	$u2$	= 2.68	[kg/(m^2 s)]

Initial temperature in the computational domain $T0$ = 2073 [K]

Geometry: `Coal.wrl`

Project: `Coal`

4.6.2.1 Physical model

In properties of the element **General settings** specify:

Reference values

Temperature **= 573** [K]

Specify modified properties of substances that are ingredients of the gas.

In the folder **Substances**:

- Create **Substance #0**.
- Load properties of **Substance #0** from the **Standard** substance database:
  - From the context menu of **Substance #0** select **Load from SD > Standard**.
  - In the **Load from database** dialog box, which opens, select:

    Substances **= Water**

    Phases **= Gas (equilibrium)**

  For the new substance **Water_Gas** specify:
  - In properties of the child element **Molar mass** specify **Value = 0.018** [kg mole^{-1}] as a constant.
  - In properties of the child element **Specific heat** specify **Value = 2000** [J/(kg·K)] as a constant.
  - In properties of the child element **Enthalpy of formation** specify **Value = -13434778** [J/kg] as a constant.
- Create another **Substance** and load it from the **Standard** substance database. In the **Load from database** dialog box, which opens, select:

  Substances **= Methane**

  Phases **= Gas**

Specify modified properties of substances that are ingredients of the coal.

For the new substance **Methane_Gas** specify:
  - In properties of the child element **Molar mass** specify **Value = 0.016** [kg mole^{-1}] as a constant.
  - In properties of the child element **Specific heat** specify **Value = 3000** [J/(kg·K)] as a constant.
  - In properties of the child element **Enthalpy of formation** specify **Value = -5502800** [J/kg] as a constant.
- Create another **Substance** and load it from the **Standard** substance database. In the **Load from database** dialog box, which opens, select:

  Substances **= Oxygen**

  Phases **= Gas**

  For the new substance **Oxygen_Gas** specify:
  - In properties of the child element **Molar mass** specify **Value = 0.032** [kg mole^{-1}] as a constant.
  - In properties of the child element **Specific heat** specify **Value = 1000** [J/(kg·K)] as a constant.
- Create another **Substance** and load it from the **Standard** substance database. In the **Load from database** dialog box, which opens, select:

  Substances **= Nitrogen**

  Phases **= Gas**

  For the new substance **Nitrogen_Gas** specify:

- In properties of the child element **Molar mass** specify **Value = 0.028** [kg mole^{-1}] as a constant.
- In properties of the child element **Specific heat** specify **Value = 1100** [J/(kg·K)] as a constant.

- Create another **Substance** and load it from the **Standard** substance database. In the **Load from database** dialog box, which opens, select:

  Substances **= Carbon dioxide**

  Phases **= Gas**

  For the new substance **Carbon dioxide_Gas** specify:
  - In properties of the child element **Molar mass** specify **Value = 0.044** [kg mole^{-1}] as a constant.
  - In properties of the child element **Specific heat** specify **Value = 1200** [J/(kg·K)] as a constant.
  - In properties of the child element **Enthalpy of formation** specify **Value = -8943409** [J/kg] as a constant.

- Create another **Substance** and load it from the **Standard** substance database. In the **Load from database** dialog box, which opens, select:

  Substances **= Water**

  Phases **= Liquid**

  For the new substance **Water_Liquid** specify:
  - In properties of the child element **Specific heat** specify **Value = 4000** [J/(kg·K)] as a constant.

- Create another **Substance** and load it from the **Standard** substance database. In the **Load from database** dialog box, which opens, select:

  Substances **= Graphite**

  Phases **= Solid**

  For the new substance **Graphite_Solid** specify:
  - In properties of the child element **Specific heat** specify **Value = 2100** [J/(kg·K)] as a constant.

- Create another **Substance** and load it from the **Standard** substance database. In the **Load from database** dialog box, which opens, select:

  Substances **= Fe2O3**

  Phases **= Solid**

  For the new substance **Fe2O3_Solid** specify:
  - In properties of the child element **Thermal conductivity** specify **Value = 1** [W/(m·K)] as a constant.
  - In properties of the child element **Specific heat** specify **Value = 1500** [J/(kg·K)] as a constant.
  - In properties of the child element **Enthalpy of formation** specify **Value = 0** [J/kg].

In the folder **Phases**:

- Create a continuous **Phase #0** (from the context menu of the folder **Phases** select the command **Create continuous**).
- Rename **Phase #0** as **Gas** (specify in its properties **Name=Gas**).
- Add the following **Substances** into the folder **Gas > Substances**:
  - **Water_Gas (equilibrium)**
  - **Methane_Gas**
  - **Oxygen_Gas**
  - **Nitrogen_Gas**
  - **Carbon dioxide_Gas**
- In properties of the folder **Gas > Physical processes** specify:

  Heat transfer **= Heat transfer via H**

  Motion **= Navier-Stokes model**

  Mass transfer **= Coal**

- In properties of the element **Gas > Physical processes > Mass transfer** specify:

i_1 *)	**= 2.6712**
i_2 *)	**= 2.1478**
i_3 *)	**= 1.5234**
Fuel	**= Methane_Gas**
Oxidizer	**= Oxygen_Gas**
Product-1	**= Carbon dioxide_Gas**
Product-2	**= Water_Gas (equilibrium)**

*) These are stoichiometric coefficients of combustion of methane in oxygen.

- Create a dispersed **Phase** of the **Particles** type (from the context menu of the folder **Phases** select the command **Create particles**). Rename this **Phase** as **Coal**.
- Add the following **Substances** into the folder **Coal > Substances**:
  - **Water_Liquid**
  - **Methane_Gas**
  - **Graphite_Solid**
  - **Fe2O3_Solid**
- In properties of the folder **Coal > Physical processes** specify:

Heat transfer	**= Convection & conduction**
Phase transfer	**= Convection & diffusion**
Motion	**= Motion**
Mass transfer	**= Coal**

- From the context menu of the folder **Coal > Size spectra** select the **Create** command. The element **Coal > Size spectra > Size spectrum #0** will appear in the project tree.
- Specify diameter of particles in the size group **[0]** of **Size spectrum #0**:

Size groups > [0] > Diam. particles= 3.1e-5 [m]

- In properties of the element **Coal > Physical processes > Mass transfer** specify:

dens.initial	**= 768.9**	Initial density of coal particles, [kg/m^3]
Composition > Water_Liquid	**= 0.022**	Mass fractions of the coal **Substances** (their sum is to be 1)
Composition > Methane_Gas	**= 0.278**	
Composition > Graphite_Solid	**= 0.552**	
Composition > Fe2O3_Solid	**= 0.148**	
A_pyr	**= 10e8**	Pyrolysis rate
C_dif	**= 10e8**	Oxygen diffusion rate
A_CO2	**= 10e8**	Char combustion rate
LastWV	**= 0.1**	Mass fraction of the rest of water and the mass fraction of the rest of volatiles to be released immediately (for one time step)

In the folder **Models**:

- Create **Model #0**.
- Add phases **Gas** and **Coal** into subfolder **Model #0 > Phases**.
- In properties of the element **Model #0 > Phase interaction > Continuum-particles** specify:

Cd **= Model1**

Nu **= Model1**

Evaporation model **= Model1**

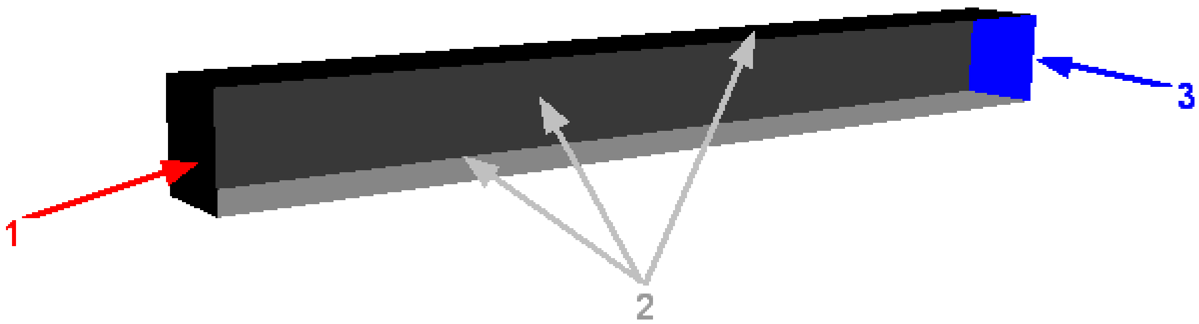
Sh **= Model1**

- For the element **Model #0 > Init. data > Init. data #0** specify:

Temperature(Gas)		
Value	**= 1500**	[K]
Velocity(Gas)		
Value > X	**= 30**	[m/s]
Mass frac. [Oxygen_Gas](Gas)		
Value	**= 0.23333**	
Velocity (disp.)(Coal)		
Value > X	**= 30**	[m/s]

In properties of **SubRegion #0**, specify: **Model = Model #0**.

4.6.2.2 Boundary conditions



Specify boundary conditions according to the illustration:

Boundary 1

Name	**= Inlet**	
Type	**= Inlet/Outlet**	
Variables		
Temperature(Gas)	**= Temperature**	
Value	**= 0**	[K]
Velocity(Gas)	**= Normal mass velocity**	
Mass velocity	**= 31.73**	[kg/(m^2·s)]
Mass frac. [Water_Gas](Gas)	**= Value at the inlet**	
Value	**= 0**	
Mass frac. [Methane_Gas](Gas)	**= Value at the inlet**	
Value	**= 0**	
Mass frac. [Oxygen_Gas](Gas)	**= Value at the inlet**	
Value	**=0.23333**	
Mass frac. [Carbon dioxide_Gas](Gas)	**= Value at the inlet**	
Value	**= 0**	
Phase volume(Coal)	**= Value**	
Value	**= 0.0003**	

Temperature (disp.)(Coal)	**= Value**	
Value	**= 0**	
Velocity (disp.)(Coal)	**= Particles mass velocity**	
Value	**= 2.68**	[kg/(m^2·s)]
Diameter(Coal)	**= Size spectrum**	
Size spectrum	**= Size spectrum #0**	
Mass frac. [Water_Liquid](Coal)	**= Coal**	
Mass frac. [Methane_Gas](Coal)	**= Coal**	
Mass frac. [Graphite_Solid](Coal)	**= Coal**	

Boundary 2

Name	**= Symmetry**
Type	**= Symmetry**

Boundary 3

Name	**= Outlet**	
Type	**= Free Outlet**	
Variables		
Temperature(Gas)	**= Temperature**	
Value	**= 0**	[K]
Velocity(Gas)	**= Total pressure**	
Total pressure	**= 0**	[Pa]
Mass frac. [Water_Gas](Gas)	**= Value**	
Value	**= 0**	
Mass frac. [Methane_Gas](Gas)	**= Value**	
Value	**= 0**	
Mass frac. [Oxygen_Gas](Gas)	**= Value**	
Value	**= 0.23333**	
Mass frac. [Carbon dioxide_Gas](Gas)	**= Value**	
Value	**= 0**	
Phase volume(Coal)	**= Permeable surface**	
Temperature (disp.)(Coal)	**= Permeable surface**	
Velocity (disp.)(Coal)	**= Permeable surface**	
Diameter(Coal)	**= Permeable surface**	
Mass frac. [Water_Liquid](Coal)	**= Permeable surface**	
Mass frac. [Methane_Gas](Coal)	**= Permeable surface**	
Mass frac. [Graphite_Solid](Coal)	**= Permeable surface**	

4.6.2.3 Initial grid

In this example, an 1D problem setting is used.

Specify in properties of the **Initial grid**:
Grid structure = 1D
Direction = X
nX = 100

In the **Properties** window of the **Initial grid** click **Apply**.

4.6.2.4 Parameters of calculation

Specify in the **Solver** tab:

- In properties of the **Time step** element specify:

Method = **In seconds**

Constant step = **0.0001** [s]

- In properties of the element **Advanced settings** specify:

Multiphase D

Cloud boundary = **0**

- In properties of the element **Limiters > Limiters for calculation > Phase limiters > Gas** specify:

Limiter

Density, min.	**= 0.01**	[kg m^{-3}]
Temperature abs, min.	**= 200**	[K]
Temperature abs, max.	**= 3000**	[K]
Velocity, max.	**= 1000**	[m s^{-1}]
Pressure abs, min.	**= 1000**	[Pa]
Pressure abs, max.	**= 300000**	[Pa]

- In properties of the element **Limiters > Limiters for calculation > Phase limiters > Coal** specify:

Limiter

Density, min.	**= 0.01**	[kg m^{-3}]
Temperature (disp.) [0], min.	**= 200**	[K]
Temperature (disp.) [0], max.	**= 3000**	[K]
Velocity, max.	**= 1000**	[m s^{-1}]

4.6.2.5 Visualization

The next sections describe how to set visualization of distribution the following values along the channel:

- mole fraction of oxygen
- mass fraction of water vapor
- temperature of the gas

Colors of axes, the plot's grid, and the plot line itself can be tuned in properties of the layers.

4.6.2.5.1 Distribution of oxygen

Create a **Line** that will be oriented along the channel. On this **Line** a **Layer** will be built displaying distribution of oxygen:

- In the **Postprocessor** tab, open the context menu of the folder **Objects** and select the **Create** command.
- In the **Create new object** dialog box, which opens, select **Object type = Line** and click **OK**.
- In properties of the new **Line #0** specify:

Object

Reference point

X = **-0.12** [m]

Z = **0.001** [m]

Create a **Layer** for visualization the distribution of oxygen:

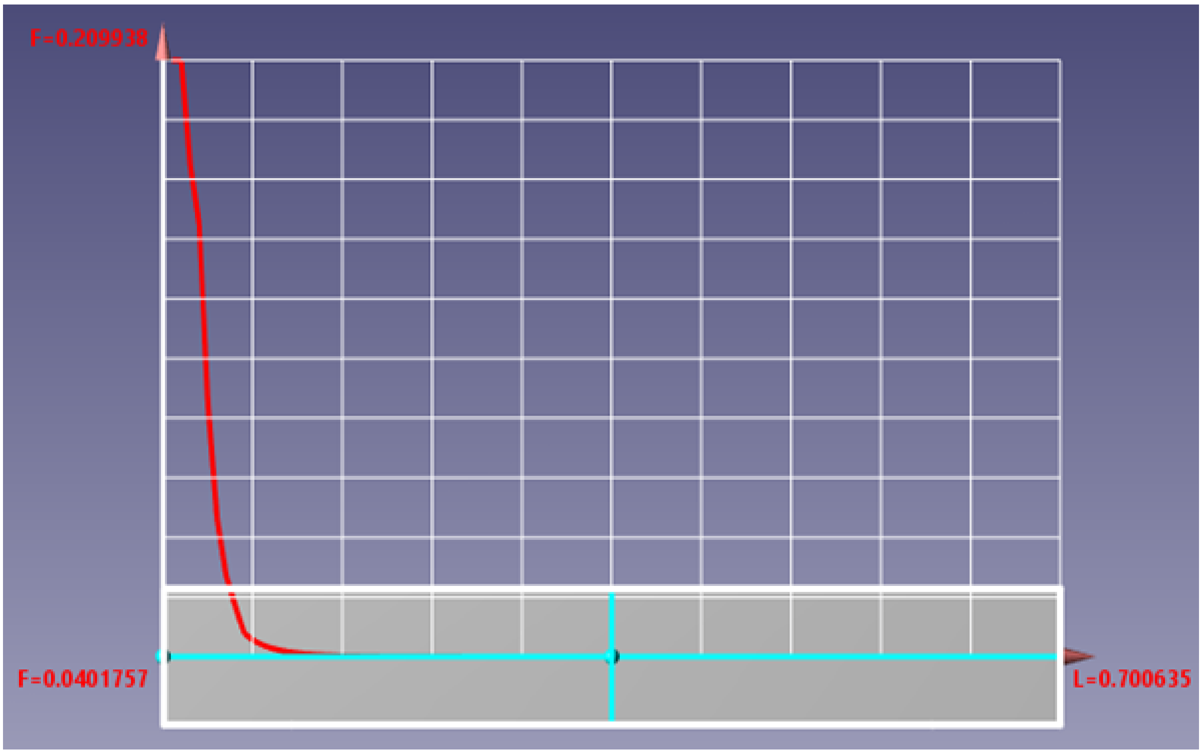
- From the context menu of **Line #0** select the **Create layer** and in the **Create new layer** dialog box, which opens, specify **Layer type = Plot along line**.
- In properties of the new line **Plot along line #0 (Line #0)** specify:

Name	**= X_O2**
Variable	
Category	**= Variables of phase "Gas"**
Variable	**= Molar frac. [Oxygen_Gas]**
Interpolation	**= No**
Number of points	**= 100**
Rotation angle	**= -90**

Run the computation and view the distribution of the oxygen mass fraction along the channel.

On the outlet of the channel the value of the oxygen mass fraction should be minimal and be near the value obtained from analytic expressions (**0.0403**).

The plot at the step number 150:



4.6.2.5.2 Distribution of water vapour

Create a **Layer** for visualization the distribution of water vapor along **Line #0**:

- From the context menu of **Line #0** select the **Create layer** and in the **Create new layer** dialog box, which opens, specify **Layer type = Plot along line**.
- In properties of the new line **Plot along line #0 (Line #0)** specify:

Name	**= X_H2O**

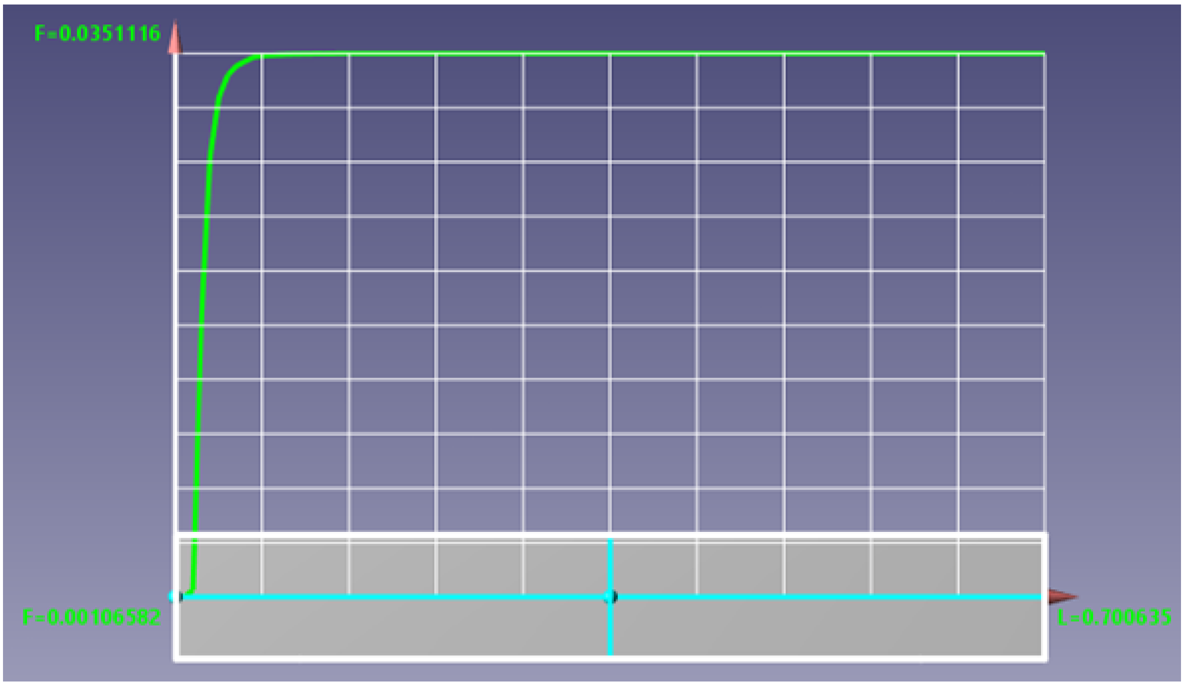
Variable

Category	**= Variables of phase "Gas"**
Variable	**= Product 2 mass frac. true** *)
Interpolation	**= No**
Number of points	**= 100**
Rotation angle	**= -90**

*) **Product 2 mass frac. true** is the true mass fraction of water vapor from the computational domain while **Mass frac. [Water_Gas (equilibrium)]** is the mass fraction of water vapor before the combustion process (recovered) that, for example, is supplied on the inlet as pure substance in the pure state; in this computation this variable refers to vapor that has evaporated from the coal.

On the outlet from the channel, value of the mass fraction of water vapor should be minimal and be near the value calculated analytically(**0.03509**).

Run the computation and view the distribution of mass fraction of water vapor along the channel.

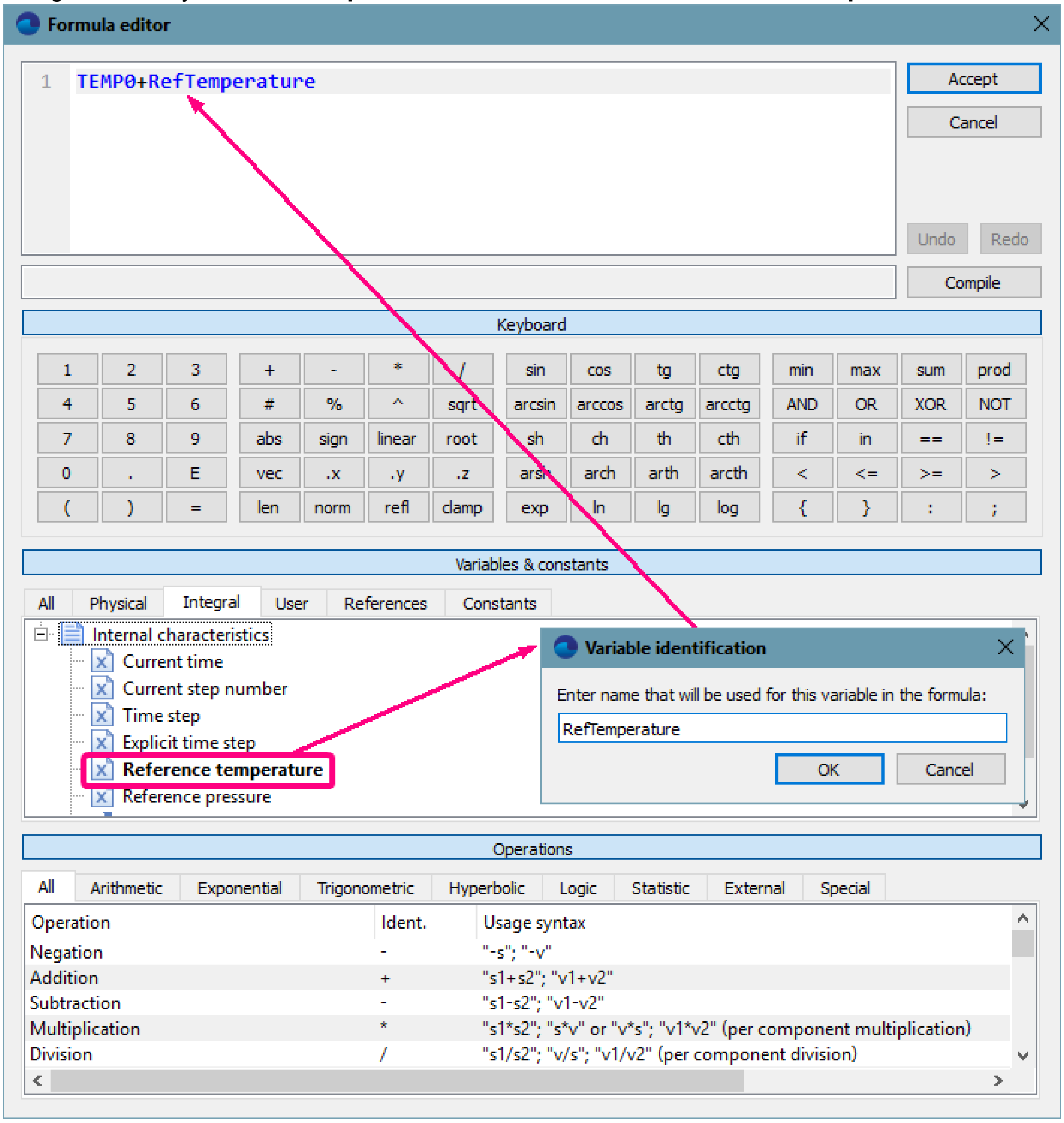


4.6.2.5.3 Distribution of the gas temperature

Create a user variable **Tgas_abs** that will correspond to temperature of the gas.

- In the **Postprocessor** tab, open the context menu of the folder **User variables > Local** and select the **Create > Scalar** command.
- A new user variable **UV #0** will appear in the folder **User variables > Local**. Specify in its properties **Name=Tgas_abs**.
- Then specify the value of the variable **Tgas_abs** using the **Formula editor**:
  - Click the icon  near the **Value** field and in the dialog box of the **Formula editor**, which opens, in the **Variables & constants** pane, double-click the variable **All > Gas > Temperature**.
  - The **Variable identification** dialog box will open where the program prompts a name, which will be used for this variable in the formula (**TEMP0**). Click **OK**. The variable identifier **TEMP0** will appear in the **Formula pane** in the upper part of the **Formula editor**.
  - In the **Formula pane** add the reference value of temperature to the existing identifier. You can do this by adding either the constant **573** or the variable **Reference temperature** from **Internal characteristics**. Let's apply the second method. Start with adding the "**+**" symbol into the **Formula pane**. Then, in the **Integral** tab double-click the **Reference temperature** variable. The **Variable**

identification dialog box will open, in which the program will prompt the standard name (**RefTemperature**) of this variable in the **Formula editor**. Click **OK** in the **Variable identification** dialog box. Finally the **Formula pane** will contain the formula **TEMP0+RefTemperature**:



- Click **Accept**.

- In the **Properties** window of the new user variable, which is being edited, click **Apply**. After this the variable will change its name to **Tgas_abs** and will have the value specified by the formula.

Create a **Layer** for visualization the distribution of the gas temperature along **Line #0**:

- From the context menu of **Line #0** select the **Create layer** and in the **Create new layer** dialog box, which opens, specify **Layer type = Plot along line**.
- In properties of the new line **Plot along line #0 (Line #0)** specify:

Name = **Tgas_abs**

Variable

Category = **User variables**

Variable = **Tgas_abs**

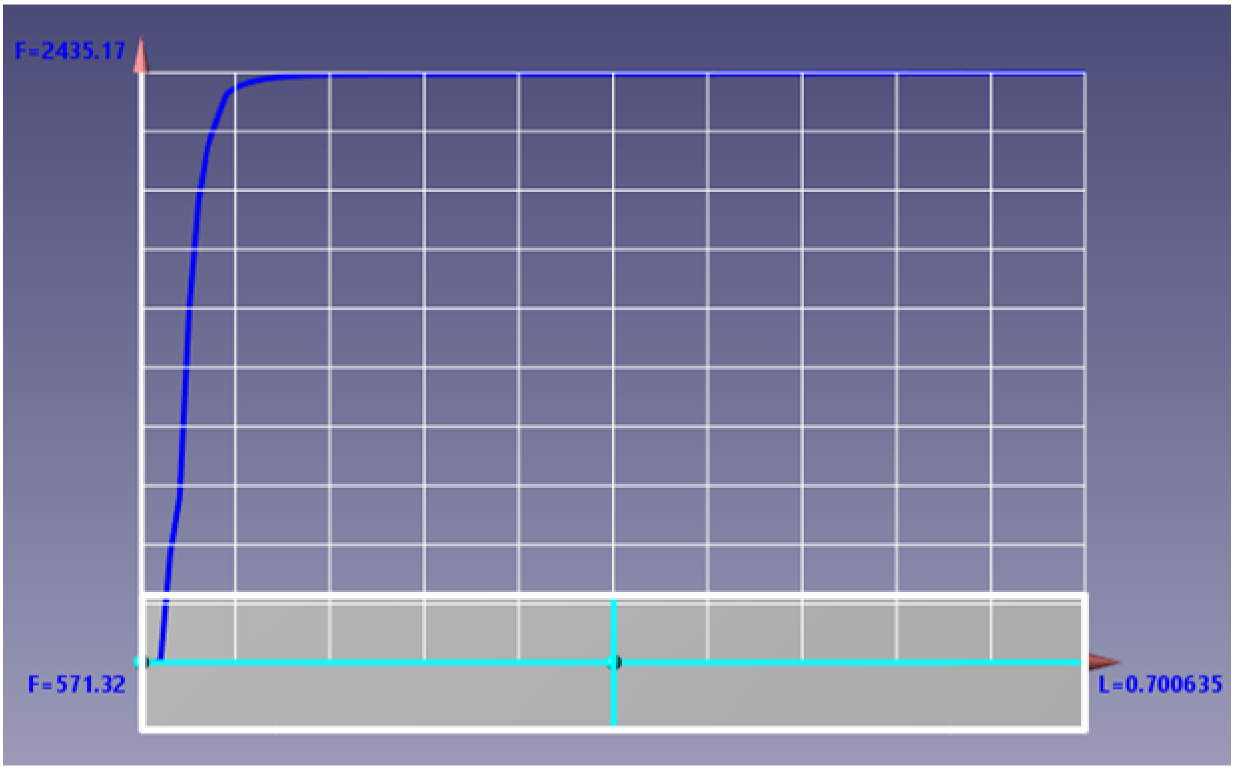
Interpolation = **No**

Number of points	**= 100**
Rotation angle	**= -90**

Run the computation and view the distribution of the gas temperature along the channel.

On the outlet of the channel the value of the gas temperature should be near to the value obtained from analytic expressions (**2440** [K]).

The plot at the step number 150:



4.7 Radiation

In *FlowVision* the following models for simulating the radiation are implemented:

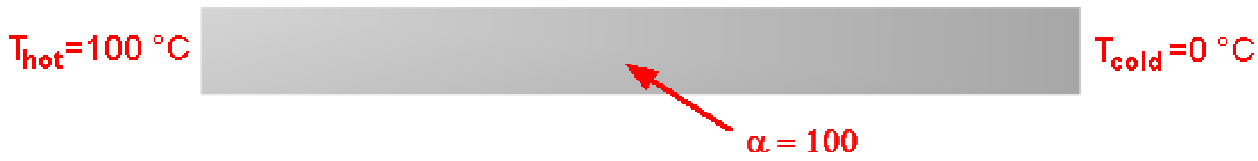
1. the radiation model P1
2. the model of an optically thin layer
3. discrete-ordinates method

In order to simulate the radiation transport, it is necessary:

- In properties of the **Substance** you have to specify the values of **Density** and **Specific heat** capacity, and, it might be necessary to specify the value of **Thermal conductivity**.
- Enable calculation of equations of **Heat transfer** and **Radiation**.
- Specify the appropriate initial and boundary conditions for the temperature and density of the radiation flux.
- If you notice some signs of instability of the solution, is recommended to reduce the time step.

4.7.1 Radiative transfer in turbid medium

Consider the one-dimensional simulation of radiation heat transfer in an opaque medium between two walls with different temperatures.



Parameters of the problem setting

Dimensions:

Length of the bar	l	= 1	[m]

Inlet parameters:

Temperature of the hot wall	T_{hot}	= 100	[K]
Temperature of the cold wall	T_{cold}	= 0	[K]
The absorption coefficient	α	= 100	[m^{-1}]

Geometry: **`Radiation.wrl`**

Project: **`Radiation`**

4.7.1.1 Physical model

In the folder **Substances**:

- Create **Substance #0**.
- Specify the following properties of **Substance #0**:

Aggregative state	**= Solid**	
Density		
Value	**= 1**	[$kg\ m^{-3}$]
Thermal conductivity		
Value	= **1e-9**	[$W\ m^{-1}K^{-1}$]
Specific heat		
Value	**= 1009**	[$J\ kg^{-1}\ K^{-1}$]

In the folder **Phases**:

- Create a continuous **Phase #0**.
- In **Phase #0** add **Substance #0** into the folder **Substances**.
- In properties of the folder **Physical processes**, specify:

Heat transfer	**= Heat transfer via h**
Radiation	**= P1**

- In properties of **Radiation**, specify:

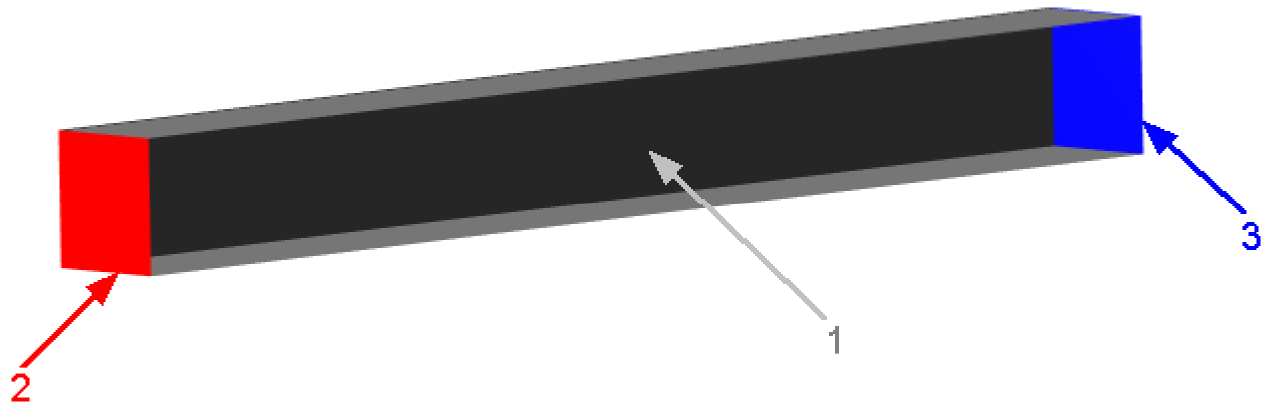
Absorption coefficient	**= 100**	[m^{-1}]

In the folder **Models**:

- Create **Model #0.**
- Add **Phase #0** into subfolder **Model #0 > Phases**.

In the **Properties** window of **SubRegion #0**, specify: **Model = Model #0**.

4.7.1.2 Boundary conditions



Specify the following boundary conditions:

Boundary 1

Type	**= Wall**	
Variables		
Temperature(Phase #0)	**= Zero gradient**	
Radiation density(Phase #0)	**= Radiation flux density**	
Value	**= 0**	

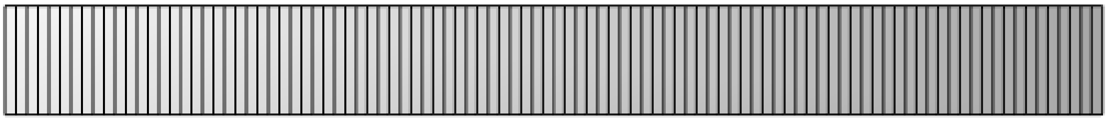
Boundary 2

Type	**= Wall**	
Variables		
Temperature(Phase #0)	**= Temperature**	
Value	**= 100**	[K]
Radiation density(Phase #0)	**= Calculating of radiation flux density**	
Blackness	**= 1**	

Boundary 3

Type	**= Wall**	
Variables		
Temperature(Phase #0)	**= Temperature**	
Value	**= 0**	[K]
Radiation density(Phase #0)	**= Calculating of radiation flux density**	
Blackness	**= 1**	

4.7.1.3 Initial grid



Specify in properties of the **Initial grid**:
Grid structure = 1D
Direction = X
nX = 100

In the **Properties** window of the **Initial grid** click **Apply**.

4.7.1.4 Parameters of calculation

Specify in the **Solver** tab in properties of the **Time step** element:

Method	**=In seconds**	
Constant step	**= 1**	[s]

Note:

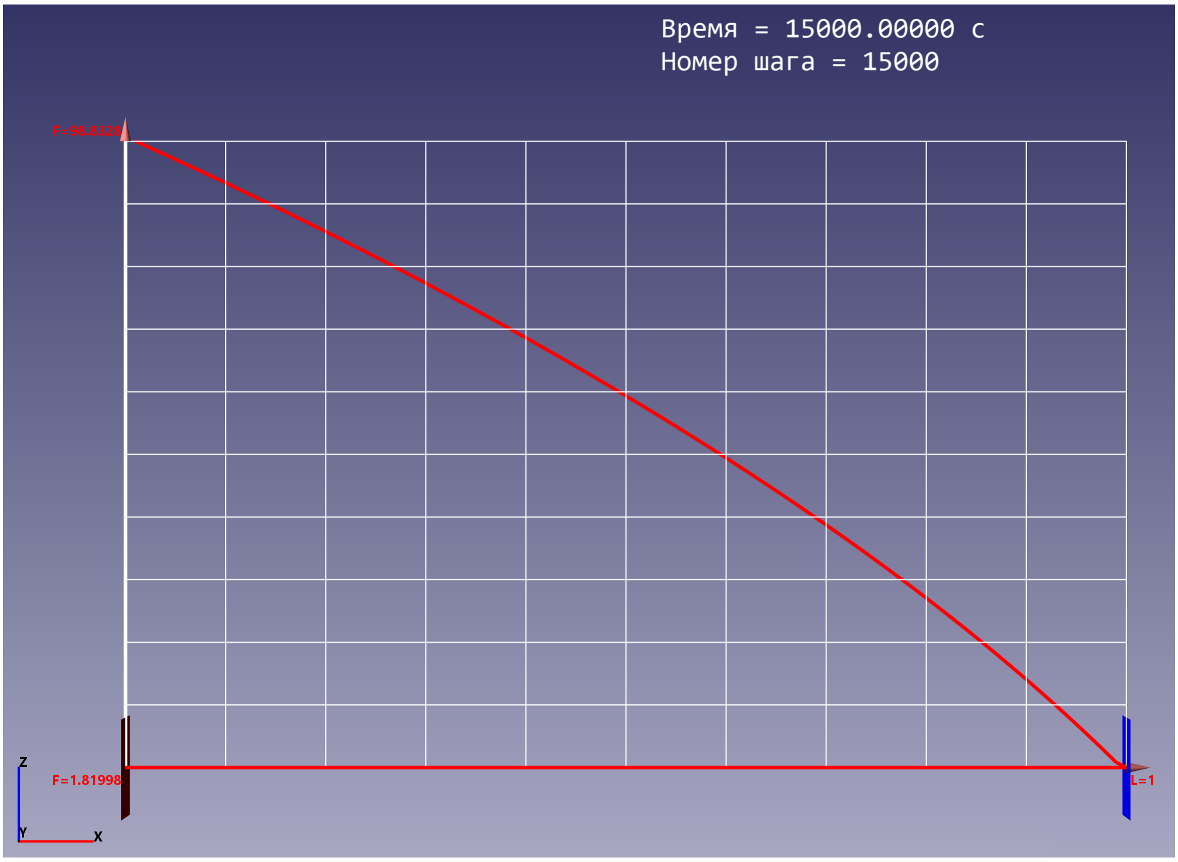
In the simulation of radiative heat transfer it is recommended to use a time step that is comparable to the diffusion CFL.

4.7.1.5 Visualization

To view the dynamics of the solution during the computation, specify visualization of the temperature distribution before the start of computation.

4.7.1.5.1 Temperature distribution

Visualization of temperature distribution when the step number is near 15000:



- Create a **Line** object in the folder **Objects**.
- In the **Properties** window of **Line #0** specify:

Object

Reference point

X	**= -0.5**	[m]
Y	**= 0**	[m]
Z	**= 0.05**	[m]

- Create a layer **Plot along line** on **Line #0**.
- In the **Properties** window of the layer **Plot along line #0 (Line #0)** specify:

Variable

Variable **= Temperature**

Number of points **= 100**

To obtain this plot, the program will require more then 10000 steps of the computation.

4.7.2 Simulating the radiative transfer by the discrete-ordinates method

The *discrete-ordinates method* is the most comprehensive model of radiation energy transfer. Implementation of the discrete-ordinates method includes angular discretization of radiative transfer equations: the whole space is split to separate solid angles with constant radiation intensity in each of them. Accuracy of the method increases with increasing the number of the solid angles.

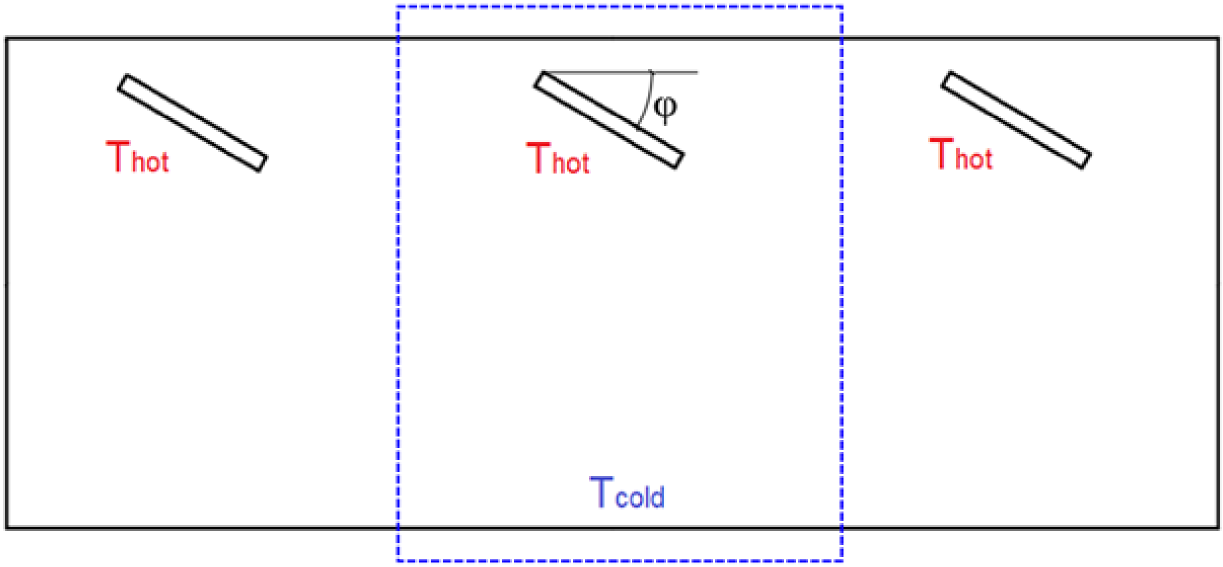
Specifics of the discrete-ordinates method is that, when curvilinear geometry is used, derivatives by angular variables in the radiative transfer equation are approximated with high accuracy.

In this training case the following situation is simulated: in a room a row of lamps are installed tilted by an angle of φ. The lamps radiate and heat the room and surfaces.

The sought-for values are:

- temperature near the lamps
- temperature near the floor

To simulate this problem setting, consider a periodic fragment of row of lamps. The fragment contains a single lamp:



This training case is simulated in 2D problem setting.

Parameters of the problem setting

Geometry parameters:

Length of the area	l	= 0.5	[m]
Width of the area	w	= 0.6	[m]
Lamp's tilt angle	φ	= 30	[degree]

Parameters:

Temperature of the hot wall	T_{hot}	= 673	[K]
Temperature of the cold wall	T_{cold}	= 273	[K]
Heat transfer coefficient of the wall	α	= 0.1	[W/(m^2·K)]

Geometry: `Plane_lamps.STL`

Project: `Lamps_MDO`

4.7.2.1 Physical model

Create a new project based on the geometry `Plane_lamps.STL`.

In the folder **Substances**:

- Create **Substance #0**.
- Load **Substance #0** from the **Standard** substance database:
  - From the context menu of **Substance #0** select **Load from SD > Standard**.
  - In the new window **Load from database**, select:

Substances	**= Air**
Phases	**= Gas (equilibrium)**

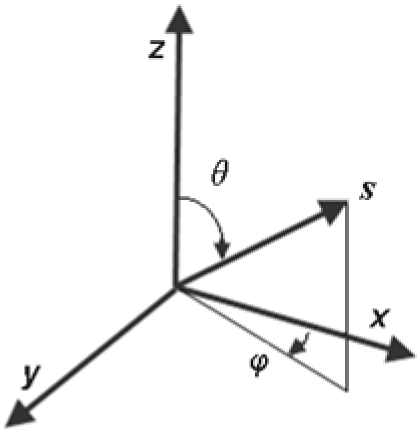
In the folder **Phases**:

- Create a continuous **Phase #0**.
- Add **Air_Gas (equilibrium)** into the subfolder **Phase #0 > Substances**.
- In properties of the folder **Physical processes**, specify:

  Heat transfer = Heat transfer via h

  Radiation = Discrete-ordinates method

Then let's specify properties of the **Radiation** physical process. The discrete-ordinates method uses discretization of the space by the polar angle θ and the azimuth angle φ of the spherical coordinate system, see the illustration below:



- In properties of **Radiation**, specify:

  Time step coefficient = 1

  Refraction index = 1
  (this value equals to the ratio of the speed of light in vacuum to the speed of light in the medium)

  Relaxation coefficient = 0.2
  (the coefficient, which is used to take into account the rate of deexcitation of atoms)

  Dispersion coefficient = 0 [m^{-1}]
  (this value equals to the ratio of the radiant flux dispersed by the body to the incident flux)

  Anisotropy dispersion coefficient = 0
  (When this coefficient of anisotropy dispersion a is non-zero, it means anisotropic emission of the radiation in the medium.
  When a>0, the radiation emits mostly in the forward direction; when a<0, the radiation emits mostly backward.)

  Number Polar Angle = 4

  Number Azimuth Angle = 8
  (this value is set by the program automatically)

  Axis for polar angle = Axis Y

- Apply the **Create** command from the context menu of the subfolder **Radiation > Spectrum** to create the element **Spectral band #0** and in properties of the element specify:

  Absorption coefficient = 1 [m^{-1}]
  (this value equals to the ratio of the radiant flux absorbed by to body to the incident flux)

The simulation will be done in the "gray" approach, so you don't have to change the default values of the **Begin wavelength** and **End wavelength** parameters.

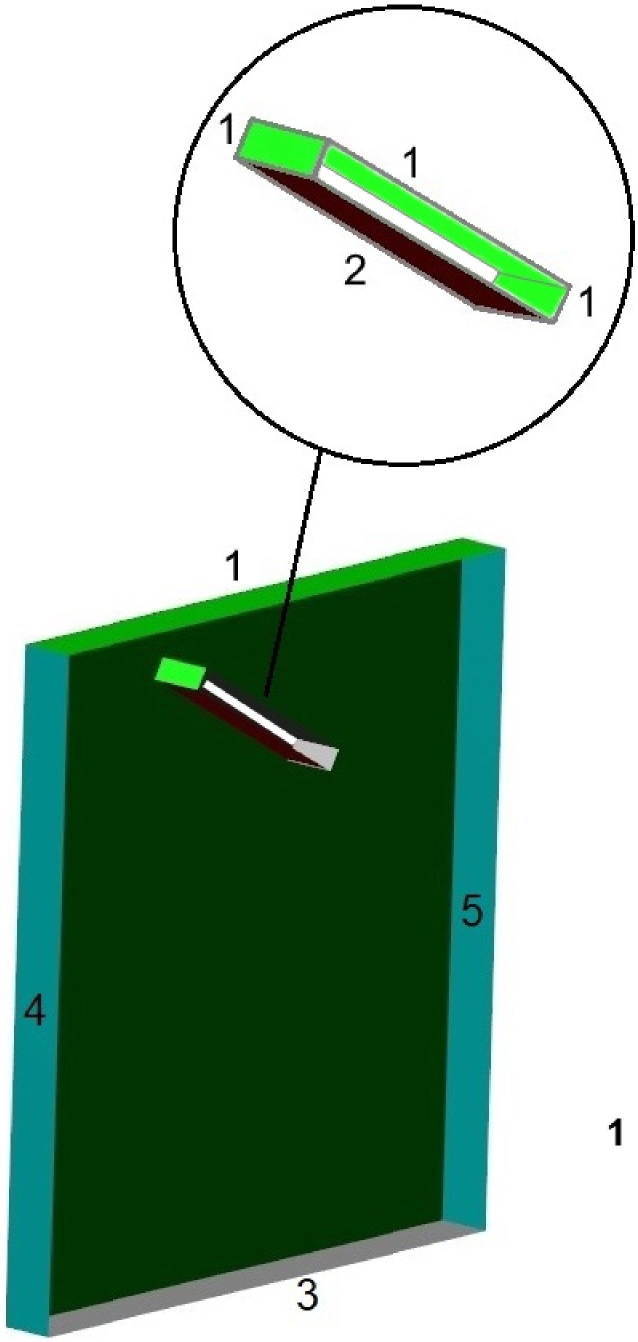
In the folder **Models**:

- Create **Model #0**.
- Add **Phase #0** into subfolder **Model #0 > Phases**.

In properties of **SubRegion #0**, specify: **Model = Model #0**.

4.7.2.2 Boundary conditions

Specify boundary conditions in **SubRegion #0** as it is shown on the illustration:



Boundary 1

Name	**= Symmetry**
Type	**= Symmetry**
Variables	
Temperature (Phase #0)	**= Symmetry**
Radiation intensity (Phase #0)	**= Opaque wall**

Boundary 2

Name	**= Hot wall**	
Type	**= Wall**	
Variables		
Temperature (Phase #0)	**= Temperature**	
Value	**= 400**	[K]
Radiation intensity (Phase #0)	**= Opaque wall**	

Boundary 3

Name	**= Cold wall**	
Type	**= Wall**	
Variables		
Temperature (Phase #0)	**= External heat exchange**	
Heat-transfer coef.	**= 0.1**	[W m^{-2} K^{-1}]
Blackness	**= 1**	
T of external medium	**= 0**	[K]
Radiation intensity (Phase #0)	**= Opaque wall**	

Boundary 4

Name	**= Left boundary**
Type	**= Connected**

Boundary 5

Name	**= Right boundary**
Type	**= Connected**

4.7.2.3 Binding the subregions

Specify snap points:

- From the context menu of the element **Boundary links > Free BCs > SubRegion #0: Left boundary** select the **Create snap point** command. The program will create the child element **SubRegion #0: Left boundary > Snap point #0**, in properties of which you have to specify:
  Coordinates
  X = 0
  Y = 0
  Z = 0
- From the context menu of the element **Boundary links > Free BCs > SubRegion #0: Right boundary** select the **Create snap point** command. The program will create the child element **SubRegion #0: Right boundary > Snap point #0**, in properties of which you have to specify:
  Coordinates
  X = 0.5
  Y = 0
  Z = 0

Create **Binder #0**:

- From the context menu of the folder **Boundary links > Binders** select the **Create all** command. The program will create the child element **Binder #0**.

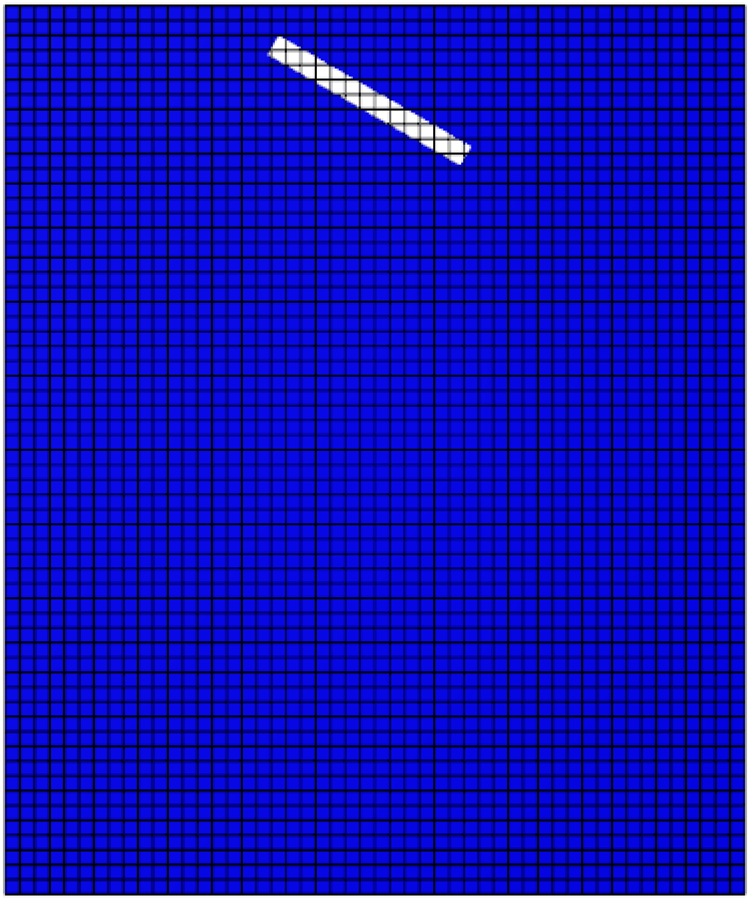
Create **Binder condition #0**:

- From the context menu of the folder **Boundary links > Binder conditions** select the **Create** command and in the **Create binder condition** dialog box, which opens, specify **Connection type = Periodic surface** (values **1st model** and **2nd model** will be already set as **Model #0**).

Specify the matching between **Binder #0** and **Binder condition #0**:

- From the context menu of the folder **Boundary links > Binder conditions > Binder condition #0 > Binders** select the **Add/Remove** command and in the **Select binders** dialog box, which opens, click the **Add All** button. Then click **OK**.
  In the project tree the element **Boundary links > Binder conditions > Binder condition #0 > Binders > Binder #0** will appear and the "**!**" mark near the element **Boundary links > Binders > Binder #0** will disappear.

4.7.2.4 Initial grid



Specify in properties of the **Initial grid**:
Grid structure = 2D
Plane = XY
nX = 50
nY = 60

In the **Properties** window of the **Initial grid** click **Apply**.

4.7.2.5 Parameters of calculation

In the **Solver** tab specify parameters of calculation.

In properties of the element **Time step** specify:

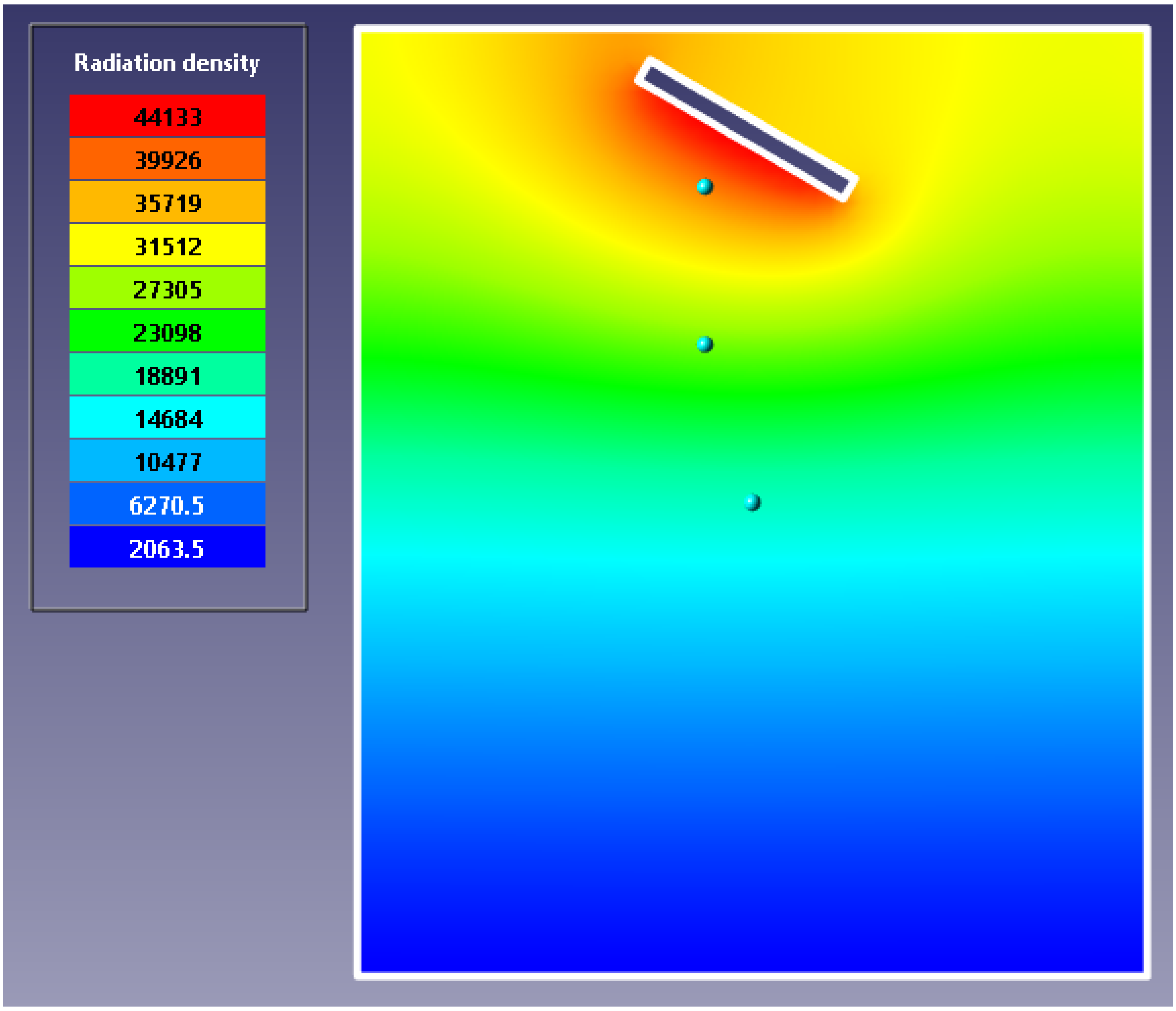
Method	**= In seconds**
Constant step	**= 100**

In properties of the element **Stopping conditions > Time steps** specify **Number = 100**.

4.7.2.6 Visualization

This training case includes the following visualizations:
- distribution of the radiation density
- distribution of temperature
- variation of temperature at a specified point

4.7.2.6.1 Distribution of the radiation density



- In properties of **Plane #0** specify:

Object

Normal

X	**= 0**
Y	**= 0**
Z	**= 1**

(to direct a **Plane**'s normal along the axis **Z**, you can also click the **Operations >** button in the **Properties** window of the **Plane**)

- On **Plane #0** create a **Color contours** layer and in its properties specify:

Name **= Radiation density**

Variable

Category **= Common and phase-unrelated variables**

Variable **= Radiation density**

Palette

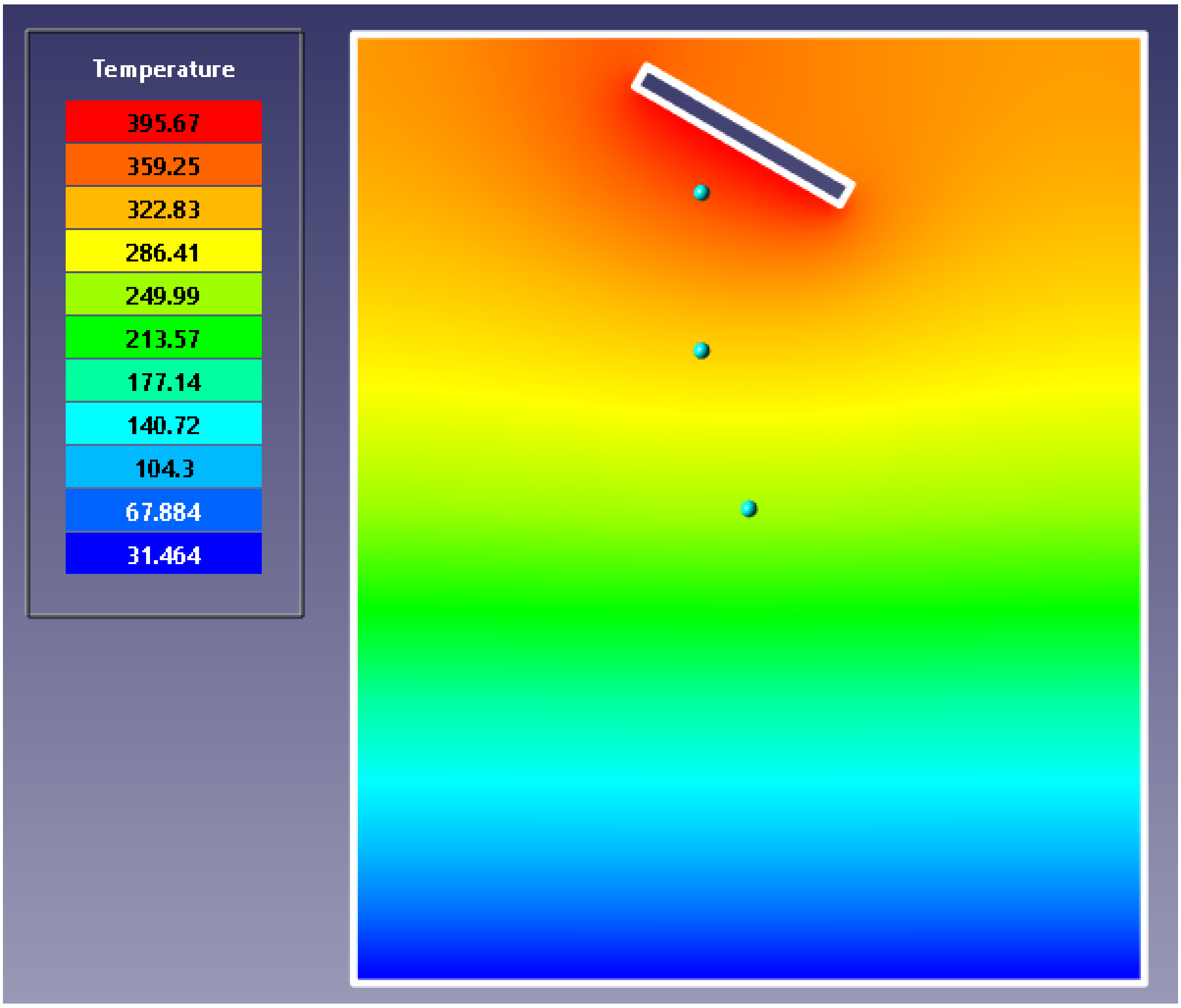
Appearance

Enabled = Yes

Title = Yes

Style = Style 1

4.7.2.6.2 Distribution of temperature



- On **Plane #0** create a **Color contours** layer and in its properties specify:

Name = **Temperature**

Variable

Category = **Common and phase-unrelated variables**

Variable = **Temperature**

Palette

Appearance

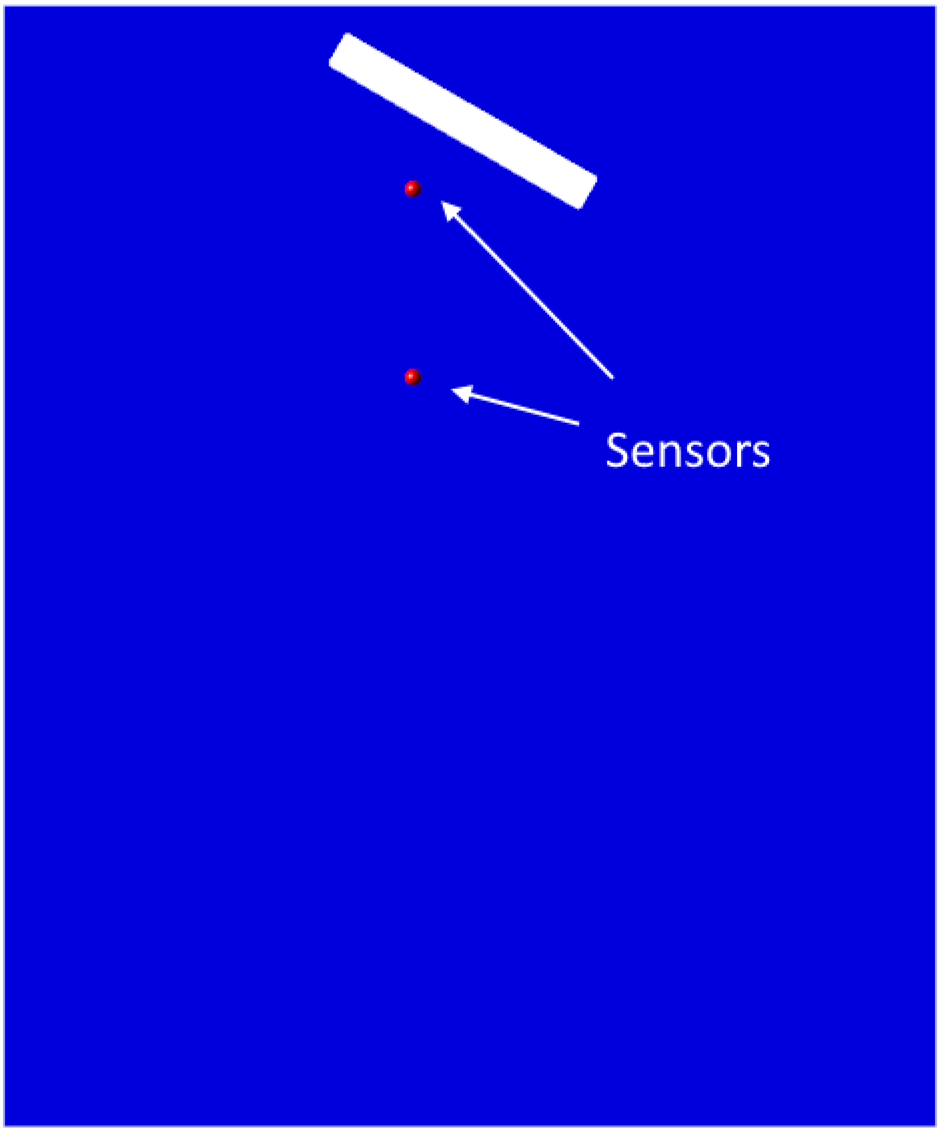
Enabled = **Yes**

Title = **Yes**

Style = **Style 1**

4.7.2.6.3 Temperature variation at a point

To obtain value of the temperature at specified points, a **Set of sensors** geometry object will be created, which contains two sensors near the lamp:



- In **Preprocessor** in the folder **Objects** create the **Set of sensors #0** object, which will include two sensors, and in its properties specify:

Sensor set

[0]

Coordinates

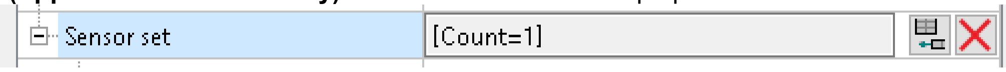
X	**= 0.22**	[m]
Y	**= 0.5**	[m]
Z	**= 0.005**	[m]

[1] *)

Coordinates

X	**= 0.22**	[m]
Y	**= 0.4**	[m]
Z	**= 0.005**	[m]

*) To create the element **[1]**, which corresponds to the second sensor, click the screen button  **(Append item to the array)** in the **Sensor set** line in properties of the **Set of sensors #0** object:



- Create a **Characteristic** on **Set of sensors #0**. In properties of the element **Characteristics #0 (Set of sensors #0)** specify:

Variable

Category	**= Common and phase-unrelated variables**
Variable	**= Temperature**

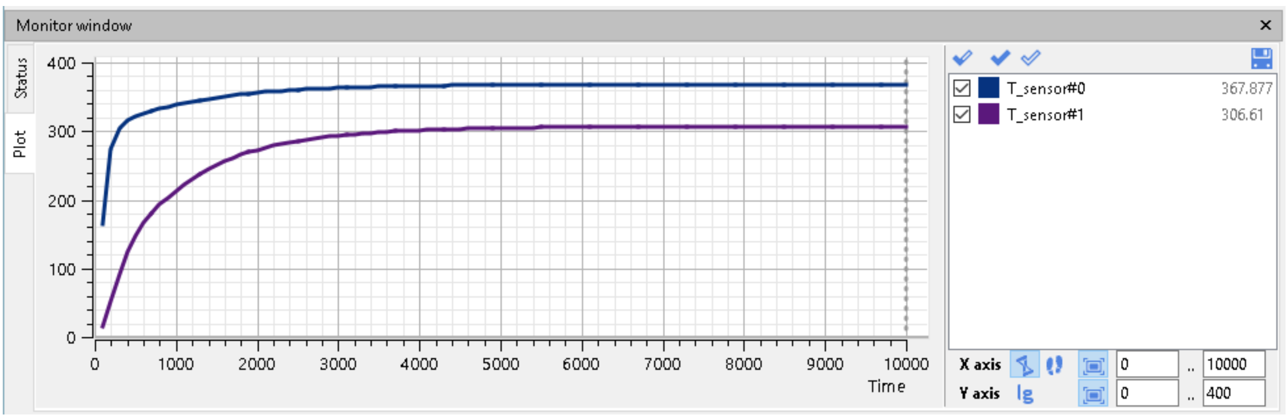
- In the **Solver** tab, in the folder **Stopping conditions > User values** create a user **Stop criterion** and in its properties specify:

Name	**= T_sensor#0**
Object	**= Characteristics #0 (Set of sensors #0)**
Group	**= Sensor #0**
Variable	**= Variable value**

- Similarly create another user **Stop criterion** with the following properties:

Name	**= T_sensor#1**
Object	**= Characteristics #0 (Set of sensors #0)**
Group	**= Sensor #1**
Variable	**= Variable value**

- Run computation of the project and view in the **Plot** tab of the **Monitor** window dynamics of **Temperature** at locations of the **Sensors**:



4.8 Electrodynamics

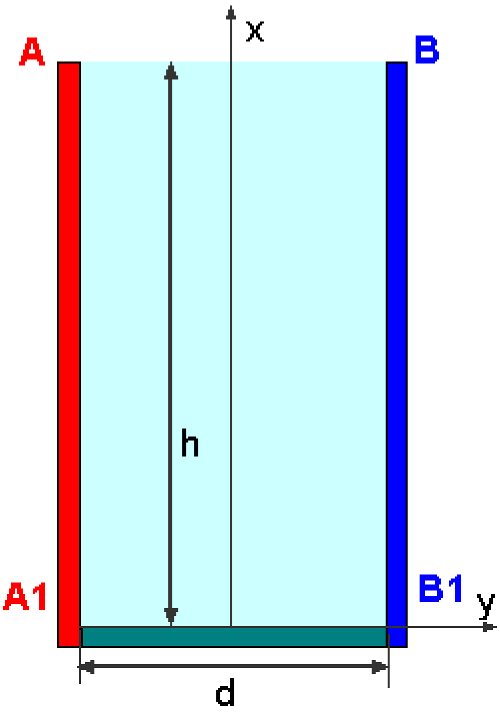
FlowVision implements simulating of electromagnetodynamics.

This is illustrated in the following exercises:

- Interaction of two isolators
- Hartmann problem

4.8.1 Interaction of two isolators

Consider the interaction of the two plates, which conduct electricity poorly. Between the upper ends of the plates (A and B) a constant voltage is maintained, while the lower ends of the plates (A1 and B1) are connected by a short-circuit conductor.



Parameters of the problem setting

Dimensions:

Height of the plates	h	= 0.02	[m]
Distance between the plates	d	= 0.005	[m]

Inlet parameters:

Voltage difference between A and B	U_{AB}	= 20	[V]
Voltage difference between A1 and B1	U_{A1B1}	= 0	[V]

Properties of the substance between isolators

Conductivity	σ	= 1	$[\Omega^{-1}m^{-1}]$
Dielectric permittivity	ε	= 1	
Relative magnetic permeability	μ	= 1	

Geometry: **`DielPlates.wrl`**

Project: **`DielPlates`**

4.8.1.1 Physical model

In the folder **Substances**:

- Create **Substance #0**.
- Specify the following properties of **Substance #0** as constants:

Aggregative state	**= Gas**	
Molar mass		
Value	**= 0.0289**	[kg $mole^{-1}$]]
Density		
Value	**= 1**	[kg m^{-3}]
Specific heat		
Value	**= 1000**	[J kg^{-1} K^{-1}]
Conductivity		
Value	**= 1**	[Ω^{-1} m^{-1}]
Permittivity		
Value	**= 1**	
Permeability		
Value	**= 1**	

In the folder **Phases**:

- Create a continuous **Phase #0.**
- Add **Substance #0** into the folder **Phase #0 > Substances**.
- In properties of **Phase #0 > Physical processes** specify:

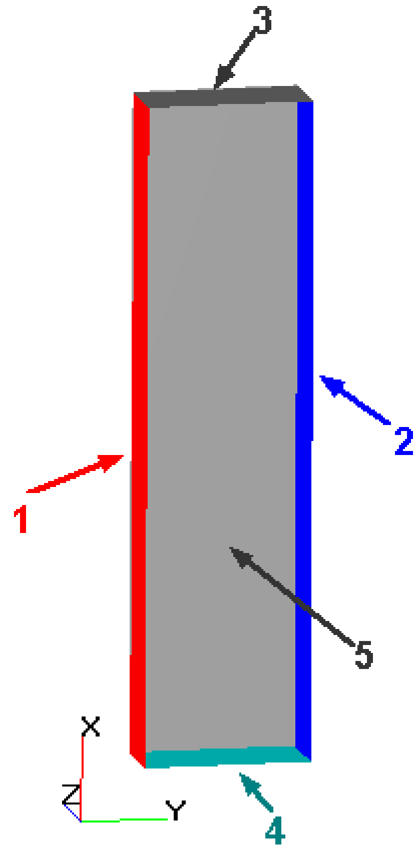
EMHD	**= Electrodynamics**

In the folder **Models**:

- Create **Model #0.**
- Add **Phase #0** into subfolder **Model #0 > Phases.**

In the **Properties** window of **SubRegion #0**, specify: **Model = Model #0.**

4.8.1.2 Boundary conditions



Specify the following boundary conditions:

Boundary 1

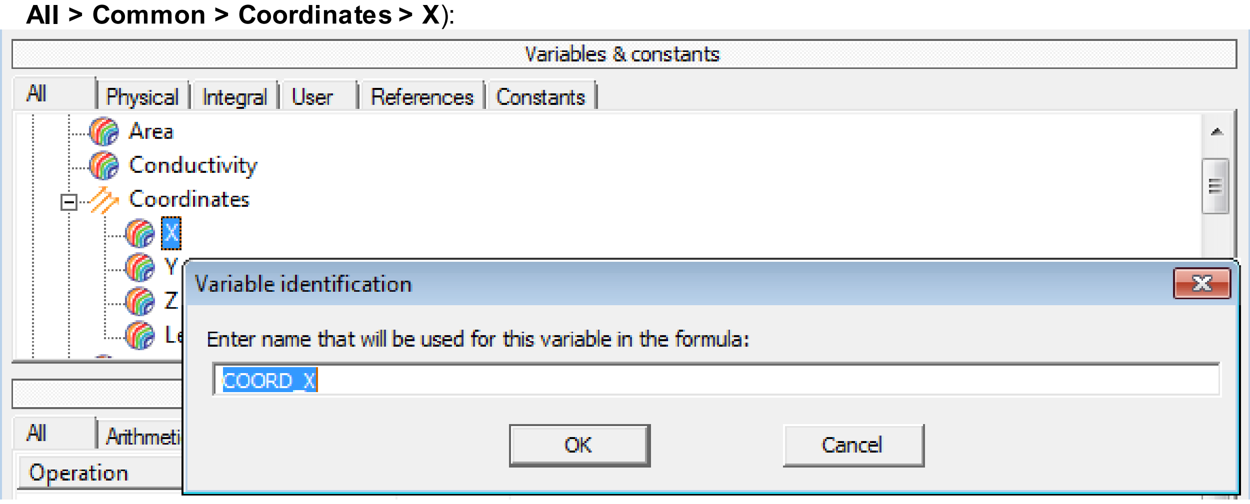
Type = **Wall**

Variables

Electrical potential (Phase #0) = **Value of el.potential**

To define the distribution of the potential on the Boundary 1, do the following:

- In properties of the element **Electrical potential (Phase #0)** change the method for specification the value in the **Value of el.potential** data field, select the **Formula** option ( ).
- Open the **Formula Editor** by clicking the icon .
- In the **Formula Editor**, create a link to the coordinate along the axis X (select **Variables & constants > All > Common > Coordinates > X**):



- Specify the linear distribution of the electric potential in the insulator between points **A** and **A1** by the formula:

$$U(x) = \frac{U_{AB}}{2} \cdot \frac{x}{h}$$

In the **Formula Editor** enter the formula: **20/2*COORD_X/0.02**

- Click the **Accept** button in the **Formula Editor**.
- Click the **Apply** button in the **Properties** window of the element **Electrical potential (Phase #0)**.

Specify the other boundary conditions. Distribution of the electric potential on boundaries 2 and 3 are respectively specified by the formulae:

between points **B** and **B1**: $$U(x) = -\frac{U_{AB}}{2} \cdot \frac{x}{h}$$

between points **A1** and **B1**: $$U(y) = -U_{AB} \cdot \frac{y}{d}$$

Boundary 2

Type	**= Wall**	
Variables		
Electrical potential (Phase #0)	**= Value of el.potential**	
Value of el.potential	**= -20/2*COORD_X/0.02**	[V]

Boundary 3

Type	**= Wall**	
Variables		
Electrical potential (Phase #0)	**= Value of el.potential**	
Value of el.potential	**= -20*COORD_Y/0.005**	[V]

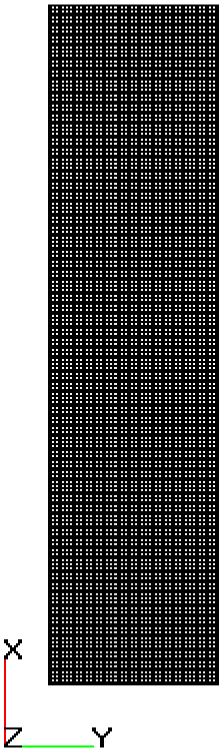
Boundary 4

Type	**= Wall**	
Variables		
Electrical potential (Phase #0)	**= Value of el.potential**	
Value of el.potential	**= 0**	[V]

Boundary 5

Type	**= Symmetry**
Variables	
Electrical potential (Phase #0)	**= Symmetry**

4.8.1.3 Initial grid



Specify in properties of the **Initial grid**:
Grid structure = 2D
Plane = XY
nX = 200
nY = 50

In the **Properties** window of the **Initial grid** click **Apply**.

4.8.1.4 Parameters of calculation

In the **Solver** tab, in properties of the **Time step** element, specify:

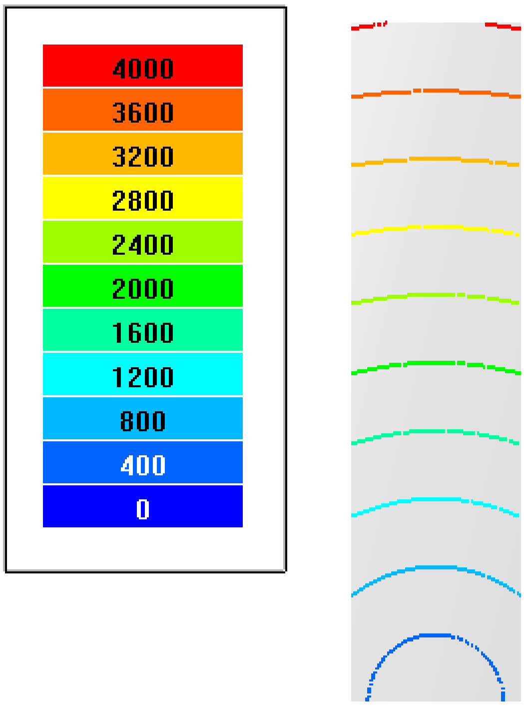
Method	**= In seconds**	
Constant step	**= 1**	[s]

4.8.1.5 Visualization

Specify visualization of:

1. **Electrical intensity's distribution** in the plane of symmetry.
2. **Electrical intensity's distribution** along the horizontal axis.

4.8.1.5.1 Electrical intensity's distribution in a plane



- In the **Properties** window of **Plane #0** specify:

Object

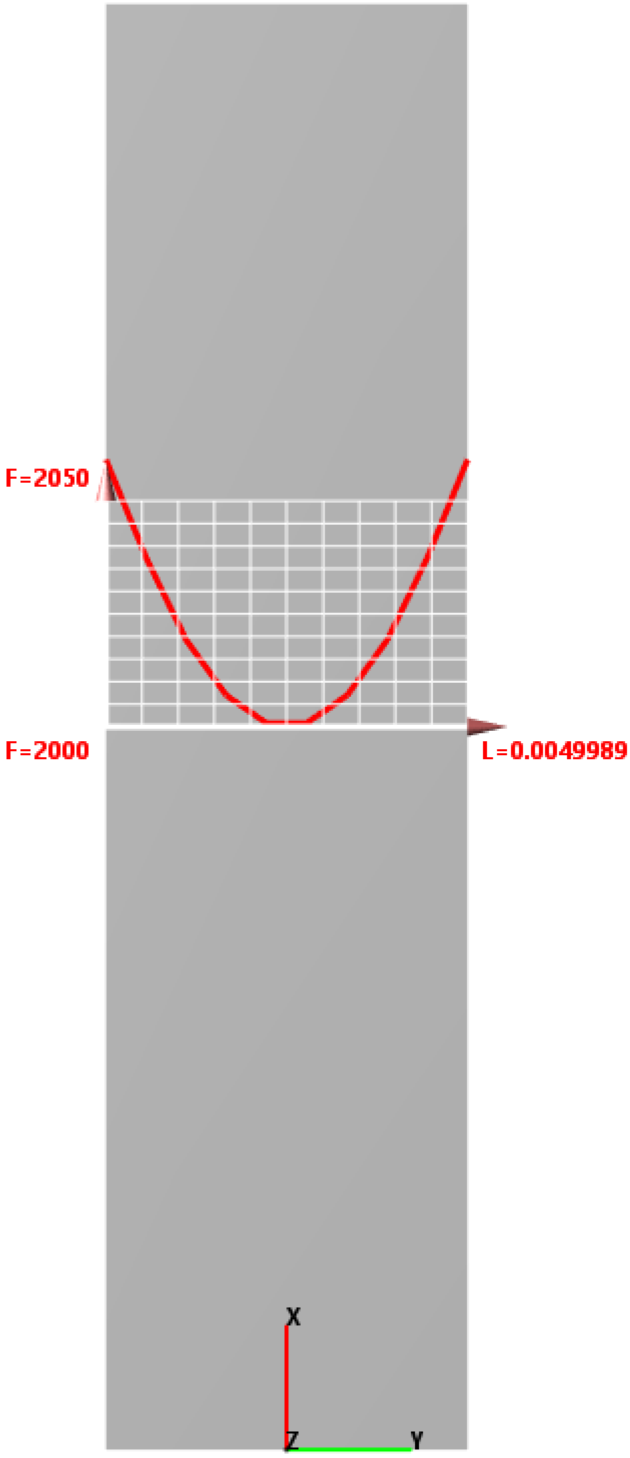
  Normal

X	**= 0**
Y	**= 0**
Z	**= 1**

- Create a layer **Color contours** on **Plane #0**.
- In the **Properties** window of the layer specify:

Variable			
	Variable		**= Electric field intensity**
Value range			
	Mode		**= Manual**
	Max		**= 4000**
	Min		**= 0**
Method			**= Isolines**
Palette			
	Appearance		
		Enabled	**= Yes**
		Style	**= Style 1**

4.8.1.5.2 Electrical intensity's distribution along a line



- Create a **Line** object in the folder **objects**
- In the**Properties** window of **Line #0**, specify:

Object

Reference point

X	**= 0.01**	[m]
Y	**= -0.002499**	[m]
Z	**= 0.00125**	[m]

Direction

X	**= 0**

Y	**= 1**
Z	**= 0**

- Create a layer **Plot along line** on **Line #0**.
- In the **Properties** window of the layer specify:

Variable	
Variable	**= Electric field intensity**
Rotation angle	**= -90**
Value range	
Mode	**= Manual**
Max	**= 2050**
Min	**= 2000**

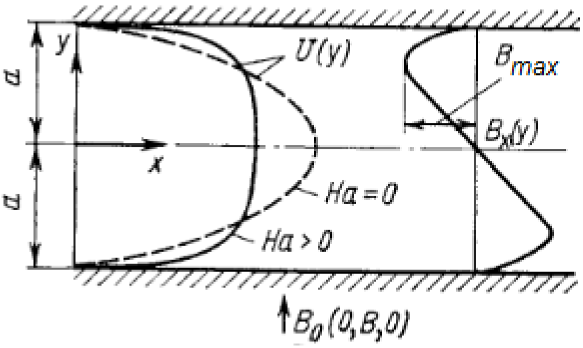
4.8.2 Hartmann problem

In this exercise we consider a laminar flow between two dielectric plates affected by external magnetic field (this is the J. Hartmann problem).

This problem has an exact analytical solution (see *Тананаев В.А. Течения в каналах МГД-устройств. - Moscow: Атомиздат, 1979, 368 pages*).

Problem setting

Incompressible liquid flows between two dielectric plates being affected by external constant transverse magnetic field B_y, with free outlet, as shown on the illustration below:



Density	ρ	= 1000	[kg/m^3]
Viscosity	μ	= 0.001	[kg m^{-1} s^{-1}]
Specific electric conductivity	σ	= 1000	[S/m]=[Ohm^{-1} m^{-1}]
Length of the channel	L	= 1	[m]
Half of the distance between the plates	a	= 0.01	[m]
Mass velocity on the inlet	ρU	= 1	[kg/(m^2 s)]
Component of the vector of the external magnetic field along axis Y	B	= 1	[T]
Hartmann number	$Ha = Ba\sqrt{\frac{\sigma}{\mu}}$	= 10	

Magnetic Reynolds number	$Re_m = \mu\,\mu_0 \cdot \sigma \cdot a\,U_0$	= 1.25664 10^{-8}
Reynolds number	$Re = \dfrac{\rho \cdot \upsilon \cdot a}{\mu}$	= 10
Relative dielectric permittivity	ε	= 1
Relative magnetic permeability	μ_r	= 1

Geometry: `Hartmann.wrl`

Project: **Hartmann**

4.8.2.1 Physical model

In the folder **Substances**:

- Create **Substance #0** with the following properties:

Aggregative state	**= Liquid**	
Molar mass		
Value	**= 0.018**	[kg $mole^{-1}$]
Density		
Value	**= 1000**	[kg m^{-3}]
Viscosity		
Value	**= 0.001**	[kg m^{-1} s^{-1}]
Thermal conductivity		
Value	**= 1e-09**	[W $(m K)^{-1}$]
Specific heat		
Value	**= 1009**	[J $(kg K)^{-1}$]
Conductivity		
Value	**= 1000**	[Ω^{-1} m^{-1}]
Permittivity		
Value	**= 1**	
Permeability		
Value	**= 1**	

In the folder **Phases**:

- Create a continuous **Phase #0**.
- Add **Substance #0** into the folder **Phase #0 > Substances**.
- In properties of **Phase #0 > Physical processes** specify:

Motion	**= Navier-Stokes model**
EMHD	**= MHD Potential model**

In the folder **Models**:

- Create **Model #0**.
- Add **Phase #0** into subfolder **Model #0 > Phases**.

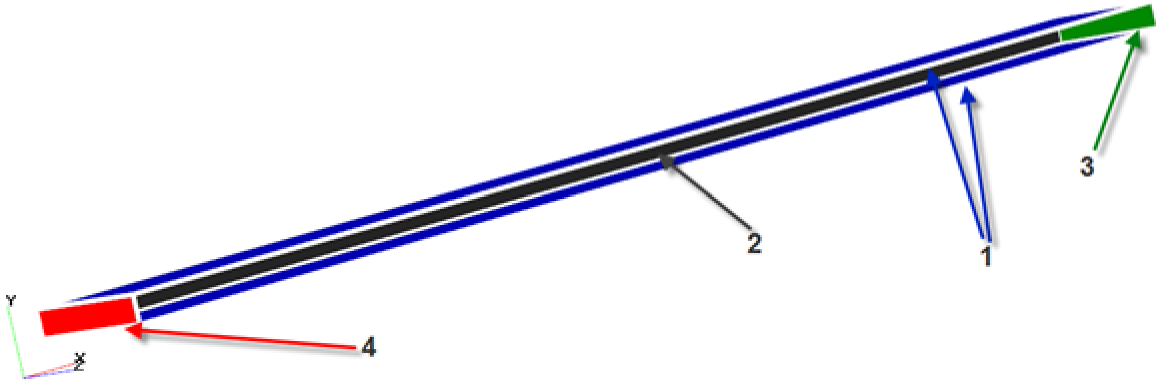
- In properties of the element **Model #0 > Init. data > Init. data #0 > Velocity (Phase #0)** specify:

Value

X	**= 0.001**	[m/s]
Y	**= 0**	[m/s]
Z	**= 0**	[m/s]

In properties of **SubRegion #0**, specify: **Model = Model #0**.

4.8.2.2 Boundary conditions



Specify the following boundary conditions:

Boundary 1

Name	**= Wall**
Type	**= Wall**
Variables *)	
Velocity (Phase #0)	**= No slip**
Electrical potential (Phase #0)	**= Zero gradient**
Magnet potential (Phase #0)	**= Zero gradient**

*) Do not change the existing values of these parameters.

Boundary 2

Name	**= Symmetry**
Type	**= Symmetry**
Variables *)	
Velocity (Phase #0)	**= Slip**
Electrical potential (Phase #0)	**= Symmetry**
Magnet potential (Phase #0)	**= Symmetry**

*) Do not change the existing values of these parameters.

Boundary 3

Name	**= Outlet**
Type	**= Free Outlet**
Variables *)	
Velocity (Phase #0)	**= Pressure**

Electrical potential (Phase #0) = **Zero gradient**

Magnet potential (Phase #0) = **Zero gradient**

*) Do not change the existing values of these parameters.

Boundary 4

Name = **Inlet**

Type = **Inlet/Outlet**

Variables

Velocity (Phase #0) = **Normal mass velocity**

Mass velocity = **1** [kg m^{-2} s^{-1}]

Electrical potential (Phase #0) = **Zero gradient**

Magnet potential (Phase #0) = **Zero gradient**

4.8.2.3 Setting the external magnetic field

The external magnetic field is set using a **Modifier** of the **External Induction** type. Follow these steps:

- Select **Create** from the context menu of the **Subregions > SubRegion #0 > Modifiers** folder.
- In the **Create new modifier** dialog box, which opens, specify:

Modifier type = **External Induction**

Objects = **Computational space**

- In properties of the **External Induction #0** modifier, which appears, specify:

Activation

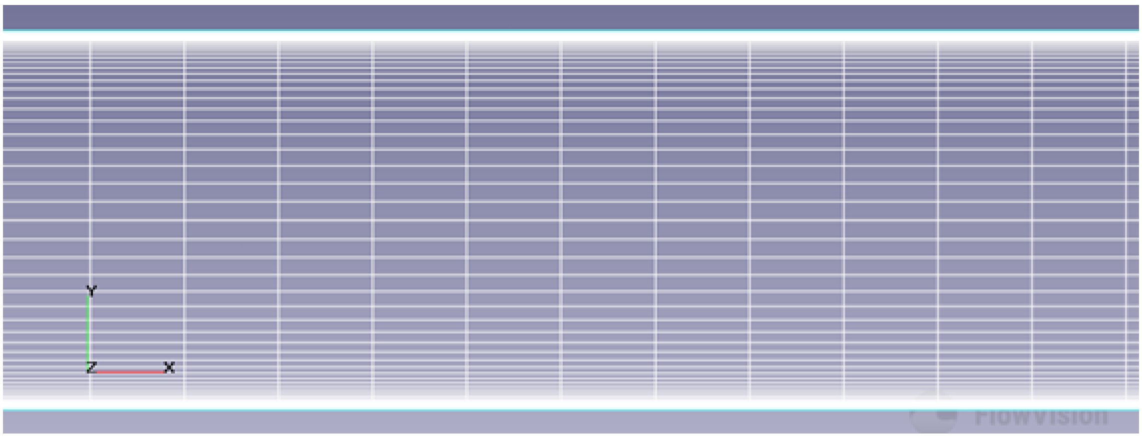
Type = **Permanent**

Induction vector

X	= **0**	[T]
Y	= **1**	[T]
Z	= **0**	[T]

4.8.2.4 Initial grid

For correct simulation of fluid friction on the wall, you have to build a two-dimensional nonuniform grid, which is condensed near the walls.



In the **Properties** window of the **Initial grid**, click the button  to open the **Initial grid editor**.

Specify in the **Initial grid editor**:

for axis OY (click the button )

Grid parameters

h_max **= 0.001** [m]

h_min **= 1e-005** [m]

Insert a reference line with coordinate **x=0** [m].

Specify **Reference line parameters** for the reference line with coordinate **x=-0.01**:

h **= 1e-005** [m]

kh+ **= 1.1485**

Specify **Reference line parameters** for the reference line with coordinate **x=0**:

h **= 0.001** [m]

kh- **= 1**

kh+ **= 1**

Specify **Reference line parameters** for the reference line with coordinate **x=0.01**:

h **= 1e-005** [m]

Click **OK** to close the **Initial grid editor** with saving the entered data.

Specify in properties of the **Initial grid**:
Grid structure = 2D
Plane = XY
nX = 200

In the **Properties** window of the **Initial grid** click **Apply**.

4.8.2.5 Parameters of calculation

Specify parameters of elements in the **Solver** tab.

- In properties of the **Time step** element, specify:

Method **= Via CFL number**

Convective CFL **= 20**

4.8.2.6 Stopping conditions

Let us create a user **Stop criterion** that is based on average pressure on the inlet. This stopping criterion will allow you to watch dynamics of pressure on the inlet that will be displayed as a plot in the **Monitor** window. Follow these steps:

- In the **Preprocessor** tab, create a **Supergroup**, selecting the **Create supergroup > In Preprocessor** command from the context menu of the boundary condition **Inlet**.
- From the context menu of the just created geometry object **Supergroup on "Inlet"** select the **Create characteristics** command.
- In properties of the just created element **Characteristics #0** specify:

Name **= Pressure (Supergroup on "Inlet")**

Variable

Variable **= Pressure**

- In the **Solver** tab, create a **Stop criterion** in the folder **Stopping conditions > User values**.
- In properties of the just created **Stop criterion #0** specify:

Name **= Aver. Pressure on inlet**

Level **= 0** *)

Object **= Pressure (Supergroup on "Inlet")**

Variable **= <f surf.>**

*) When you set **Level = 0**, the computation will not stop by this criterion but the **Plot** tab of the **Monitor** will display the plot of the specified variable over time.

Also add a stopping condition based on number of steps. To do so, specify **Number =30** in properties of the element **Stopping conditions > Time steps**.

4.8.2.7 Visualization

- Profiles of velocity and magnetic induction
- Variation of pressure on inlet

4.8.2.7.1 Profiles of velocity and magnetic induction

To visualize profiles of the velocity and magnetic induction, two **Plot along line** layers will be created. Also a **Vectors** layer will be created to visualize the velocity.

In the **Postprocessor** tab, create **Planes** for the visualization:

- From the context menu of the **Objects** folder, select **Create**.
- In the **Create new object** dialog box, which opens, select **Object type = Plane**.
- In properties of the just created **Plane #1** specify:

Object

Reference point

X	**= 0.94**
Y	**= -0.01**
Z	**= -0.05**

Normal

X	**= 0**
Y	**= 0**
Z	**= -1**

- Create also **Plane #2** with the following properties:

Object

Reference point

X	**= 0.97**
Y	**= -0.01**
Z	**= -0.05**

Normal

X	**= 0**
Y	**= 0**
Z	**= -1**

To visualize profiles of magnetic induction, create a **Plot along line** layer on **Plane #1**:

- In the **Postprocessor** tab, create a **Plot along line** layer on **Plane #1**. To do so, select the **Create layer** command from the context menu of **Plane #1** and in the **Create new layer** dialog box, which opens, specify **Layer type = Plot along line** and **Objects = Plane #1**.
- In properties of the just created layer **Plot along line #0** specify:

Name	**= Induction**
Variable	
Variable	**= Induction**

Component	**= X**
Number of points	**= 100**
Value range	
Mode	**= Manual**
Max	**= 9.7e-10**
Min	**= -9.7e-10**
Appearance	
Plot	
Color	**= Red**
Width	**=4**

* On **Plane #2** create **Plot along line #1** layer with the following properties:

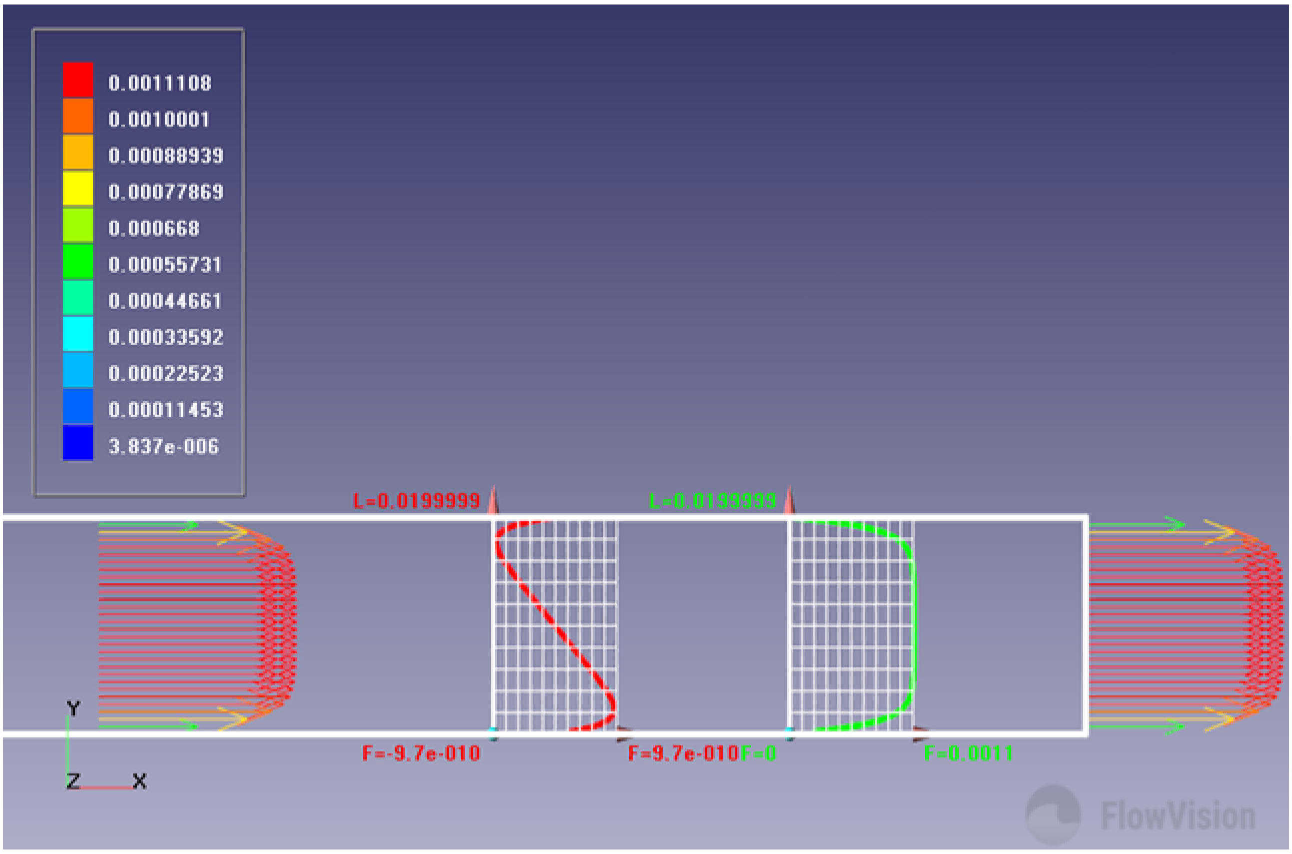
Name	**= Velocity**
Variable	
Variable	**= Velocity**
Component	**= X**
Number of points	**= 100**
Value range	
Mode	**= Manual**
Max	**= 0.0011**
Min	**= 0**
Appearance	
Plot	
Color	**= Green**
Width	**=4**

* On **Plane #1** create a **Vectors** layer with the following properties:

Name	**= Velocity vectors**
Variable	
Variable	**= Velocity***)
Grid	
Size 2	**= 30**
Coloring**)	
Variable	
Variable	**= Velocity**
Palette	
Appearance	
Enabled	**= Yes**

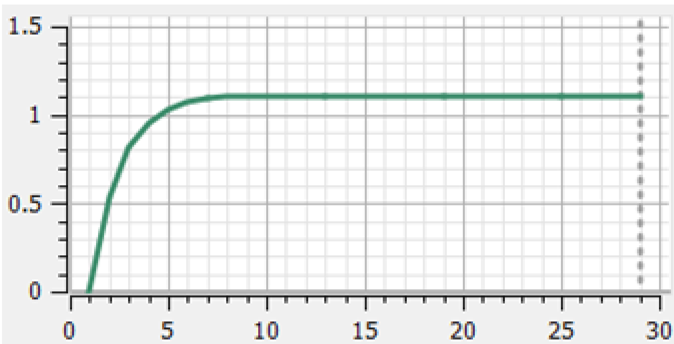
*) For **Vectors** layers the program by default sets **Variable > Variable = Velocity**.
**) These parameters are set to color velocity vectors depending on their absolute values.

As a result, profile of velocity, profile of magnetic induction, and vectors of velocity will be displayed during the computation (at step number 30):



4.8.2.7.2 Variation of pressure on inlet

Variation of the average pressure on the inlet during the computation will be displayed on the **Plot** tab in the **Monitor** window:



5 Advanced modules

This section describes the following additional features of *FlowVision*:

1. **Conjugate simulations**
2. **Rotation**
3. **Moving bodies**
4. **Icing on a solid surface**

5.1 Conjugate simulation

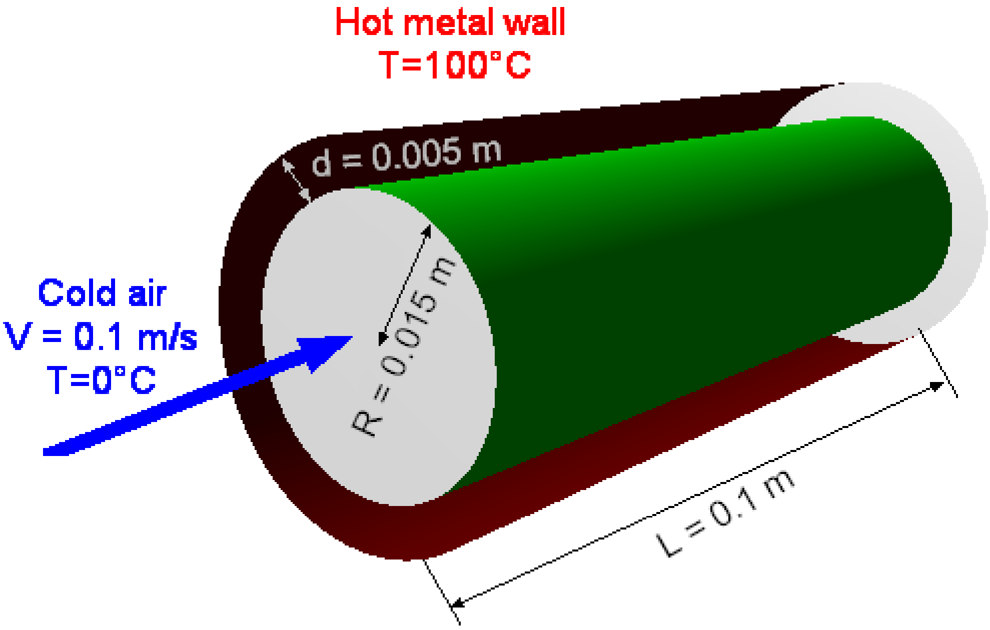
Conjugate simulations take into account the interaction between the different **Subregions** or different boundaries of one **Subregion**.

In order to run conjugate simulations, you have to:

- Prepare a geometric model of the computational domain, consisting of several **Subregions**.
- Set the boundary condition **Connected** on both sides of surfaces, on which the subregions' conjugation will be calculated.
- Create appropriate **Binders**, **Binder conditions** and matching between these **Binders** and **Binder conditions**.

5.1.1 Conjugate heat exchange

Consider a simulation of a laminar flow of viscous cold air in a tube with a thick metal wall heated from the outside.



Parameters of the problem setting

Dimensions:

Length of the tube	L	= 0.1	[m]
The inner radius of the tube	R	= 0.015	[m]
Thickness of the wall	d	= 0.005	[m]

Inflow parameters:

Velocity on inlet	V_{inl}	= 0.1	[m s^{-1}]
Temperature on inlet	T_{inl}	= 0	[K]
Temperature of the outer surface of the tube	T_w	= 100	[K]

Fluid parameters:

Density	ρ	= 1	[kg m^{-3}]
Viscosity	μ	=1.82e-5	[kg m^{-1} s^{-1}]
Thermal conductivity	λ	= 0.026	[W m^{-1} K^{-1}]

Specific heat	c_p	= 1009	[J kg^{-1} K^{-1}]

The parameters of the solid substance:

Density	ρ	= 7900	[kg m^{-3}]
Thermal conductivity	λ	= 45	[W m^{-1} K^{-1}]
Heat capacity	c_p	= 457	[J kg^{-1} K^{-1}]

Reynolds number:

$$Re = \frac{2RV_{inl}\rho}{\mu} = \frac{2 \cdot 0.015 \cdot 0.1 \cdot 1}{1.82 \cdot 10^{-5}} \approx 165$$

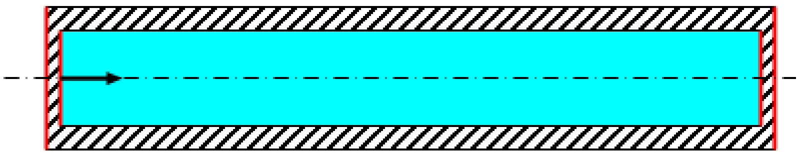
5.1.1.1 Making the project based on a single detail

Geometry: `Conjugate_Convection.STL`

Project: `Conjugate_Convection`

5.1.1.1.1 Computational domain

When preparing a geometric model based on a single detail, a situation of so-called T-shaped surfaces arise.



To solve this problem, it is necessary to change the geometry by adding thin walls instead of the T-shaped surfaces. On the boundary of these fictitious walls the adiabatic boundary conditions are set. The disadvantage of this method is that it distorts the image of the actual physical process. If the energy equation is solved in the simulation, after introduction of such walls thermal bridges will appear.

A fully prepared geometry of the computational domain is stored in the file `Conjugate_Convection.STL`.

5.1.1.1.2 Physical model

In the folder **Substances**:

- Create **Substance #0**.
- Specify the following properties of **Substance #0**:

Name	**= Steel**	
Aggregative state	**= Solid**	
Molar mass		
Value	**= 0.056**	[kg mole^{-1}]
Density		
Value	**= 7900**	[kg m^{-3}]
Thermal conductivity		
Value	**= 45**	[W m^{-1}K^{-1}]
Specific heat		
Value	**= 457**	[J kg^{-1} K^{-1}]

- Create **Substance #0**.
- Specify the following properties of **Substance #0**:

Name	**= Air**
Aggregative state	**= Gas**
Molar mass	

Value	**= 0.0289**	[kg $mole^{-1}$]
Density		
Value	**= 1**	[kg m^{-3}]
Viscosity		
Value	**= 1.82e-5** *)	[kg $m^{-1}s^{-1}$]
Thermal conductivity		
Value	**= 0.026**	[W $m^{-1}K^{-1}$]
Specific heat		
Value	**= 1009**	[J kg^{-1} K^{-1}]

*) **1.82e-5** is notation for 1.82×10^{-5}.

In the folder **Phases**:

- Create a continuous **Phase #0**.
- In the **Properties** window of **Phase #0**, specify:

Name **= Steel**

- In the folder **Substances** of the phase **Steel** load the substance **Steel**.
- In the **Properties** window of the folder **Physical processes**, specify:

Heat transfer **= Heat transfer via h**

- Create a continuous **Phase #0**.
- In the **Properties** window of **Phase #0**, specify:

Name **= Air**

- In the folder **Substances** of the phase **Air** load the substance **Air**.
- Specify in the **Properties** window of the folder **Physical processes**:

Heat transfer **= Heat transfer via h**
Motion **= Navier-Stokes model**

In the folder **Models**:

- Create **Model #0**.
- In the **Properties** window of **Model #0**, specify:

Name **= Steel**

- Add the phase **Steel** into the folder **Models > Steel > Phases**.
- Create **Model #0**.
- In the **Properties** window of **Model #0**, specify:

Name **= Air**

- Add the phase **Air** into the folder **Models > Air > Phases**.
- In **Models > Air > Init. data > Init. data #0** specify:

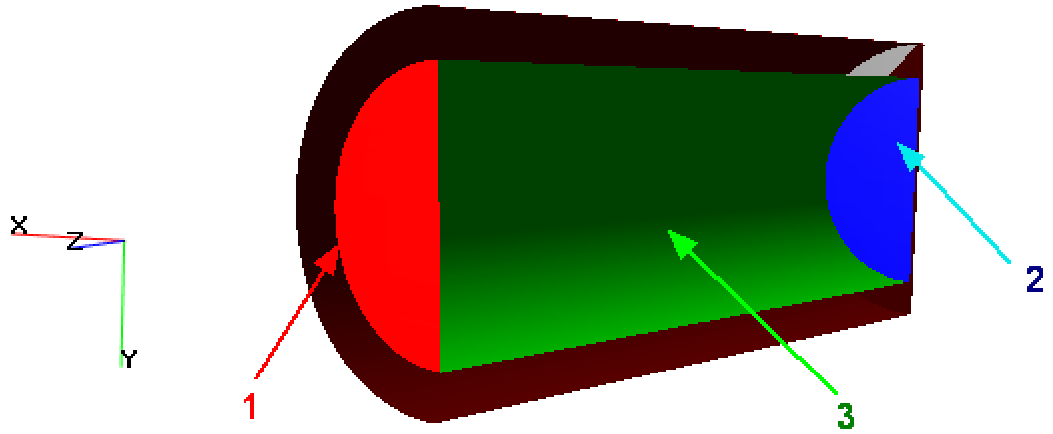
Velocity (Air)		
X	**= 0**	[m s^{-1}]
Y	**= 0**	[m s^{-1}]
Z	**= -0.1** *)	[m s^{-1}]

*) Negative value of the initial velocity is specified because the flow goes against the axis Z.

5.1.1.1.3 Boundary conditions

In the **Properties** window of the **Subregion** within the inner tube specify:

Name	**= Flow**
Model	**= Air**



Specify the following boundary conditions on the inner surface of the inner tube:

Boundary 1

Name	**= Inlet**	
Type	**= Inlet/Outlet**	
Color	**= Red**	
Variables		
Temperature(air)	**= Temperature**	
Value	**= 0**	[K]
Velocity(Air)	**= Normal mass velocity**	
Mass velocity	**= 0.1**	[kg $(m^2s)^{-1}$]

Boundary 2

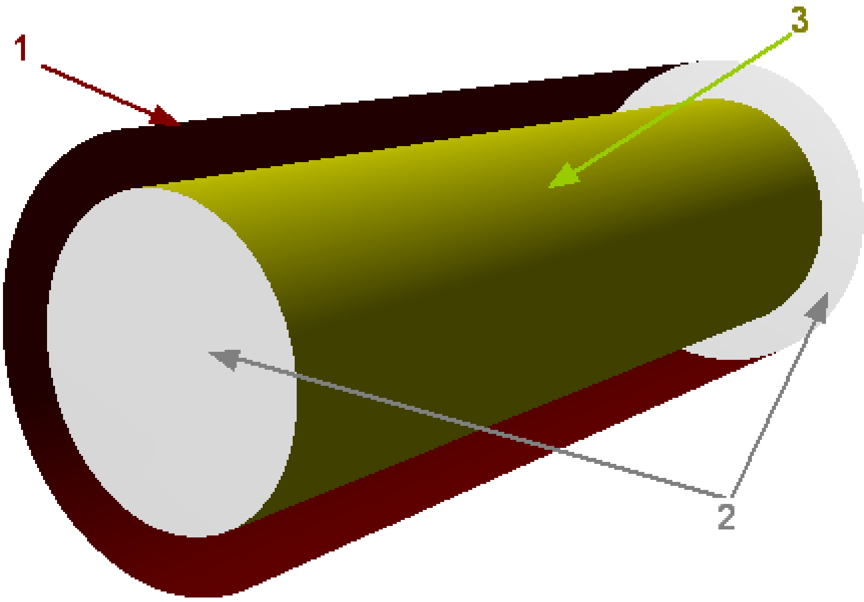
Name	**= Outlet**	
Type	**= Free Outlet**	
Color	**= Blue**	
Variables		
Temperature(air)	**= Zero gradient**	
Velocity(Air)	**= Pressure**	
Value	**= 0**	[Pa]

Boundary 3

Name	**= Wall**
Type	**= Connected**
Color	**= Green**

In the **Properties** window of the **Subregion** between the outer and inner tubes specify:

Name	**= Tube**
Model	**= Steel**



Specify the following boundary conditions on the surfaces, which are boundaries for the steel:

Boundary 1

Name	**= Outer wall**	
Type	**= Wall**	
Color	**= Maroon**	
Variables		
Temperature(Steel)	**= Temperature**	
Value	**= 100**	[K]

Boundary 2

Name	**= Wall**
Type	**= Wall**
Color	**= Gray**
Variables	
Temperature(Steel)	**= Zero gradient**

Boundary 3

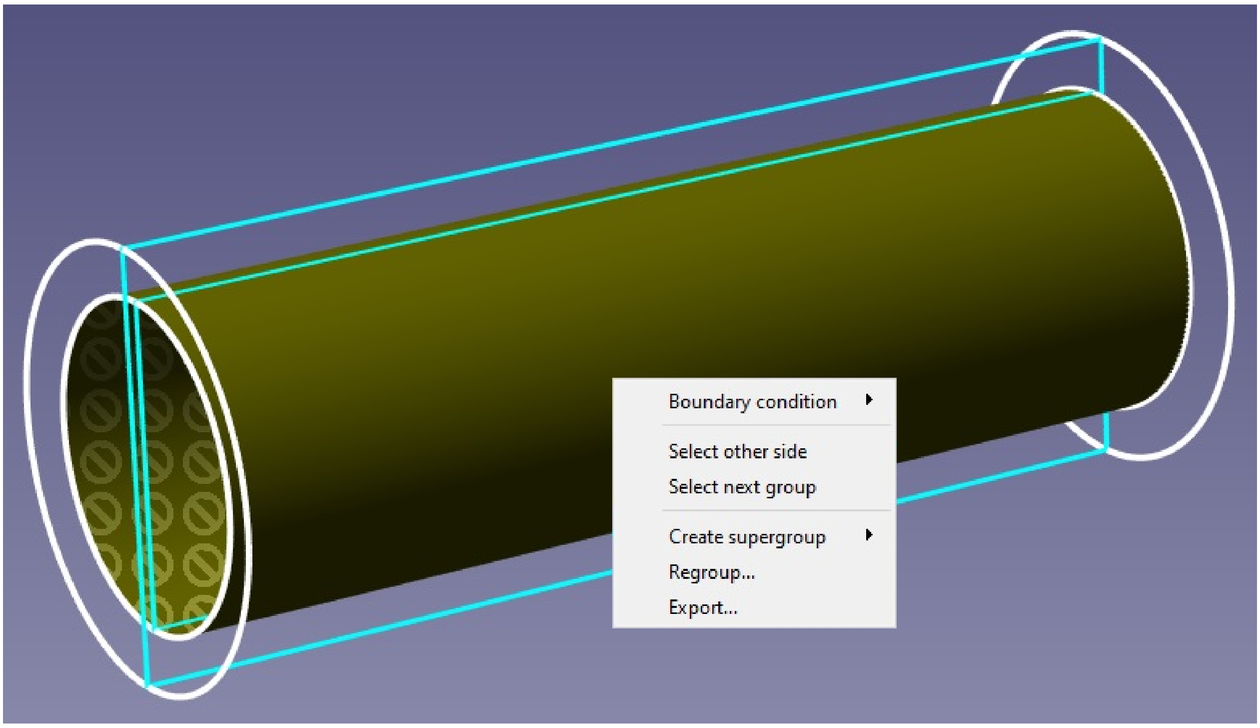
Name	**= Inner wall**
Type	**= Connected**
Color	= **Yellow**

Notes:

When setting the boundary conditions on the inner you have select not only the surface but also the side of the surface. On the outer surfaces of the inner tube the boundary conditions are specified in the subregion **Tube**, while on the inner surfaces of the inner tube the boundary conditions are specified in the subregion **Flow**. To select a side of the surface of the inner tube, to the following in the **View** window:

- Go to the **Selection mode** to select a surface in the **View** window by clicking the button (**Selection mode**) in the **Work modes** toolbar, or press and hold down the **Alt** button.
- Hover your mouse pointer over the required surface.
- Click by the left mouse button.

- If a wrong surface would be selected, click by the left mouse button repeatedly until until the required surface is selected.
- If a wrong side would be selected, then right-click the selected surface. A context menu will open, select there the option **Select other side**:



To understand, which side is selected, you can look at the pattern on its image. The selected side surface has no pattern on it, while the unselected side is marked by a pattern consisting of the "no" symbols .

Also, to select a surface, you can use the elements **Subregion > SubRegion #N > Geometry > Region - Surface #N > Region - Group #N** in the project tree in **Preprocessor**. Displaying of all geometrical groups as elements in the project tree is set by the menu command **File > Preferences** by the setting **Display > Show all groups = Yes | No**. When you have many nested subregions and geometry groups, this method becomes preferable.

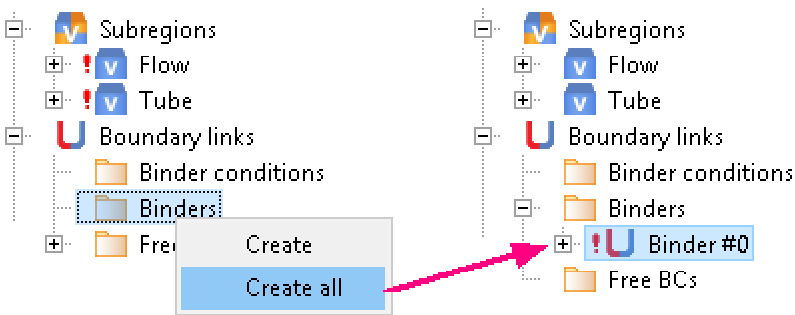
5.1.1.1.4 Binding the subregions

Binding the boundary conditions consists of the following steps:

- creating a **Binder** of two **Boundary conditions**, which types are **Connected**
- creating a **Binder condition**
- matching the **Binder** and the **Binder condition**

Create **Binder #0**:

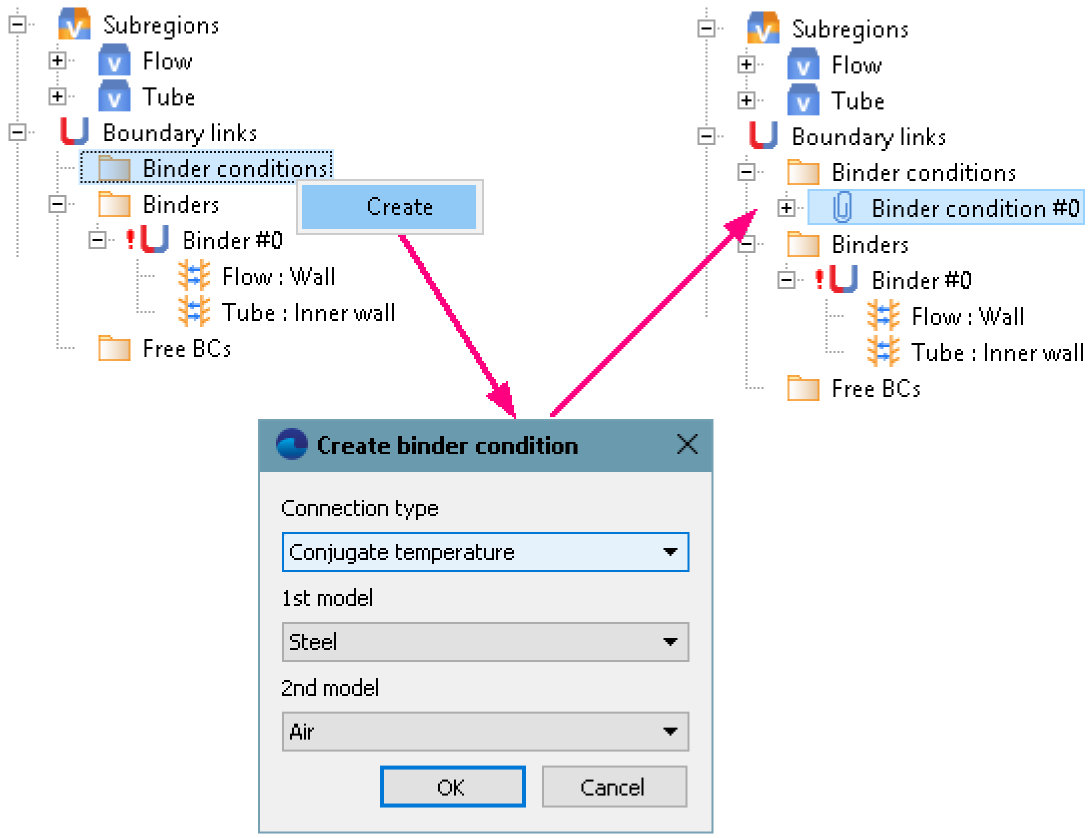
- Select **Create all** from the context menu of the folder **Boundary links > Binders**:



Create **Binder condition #0**:

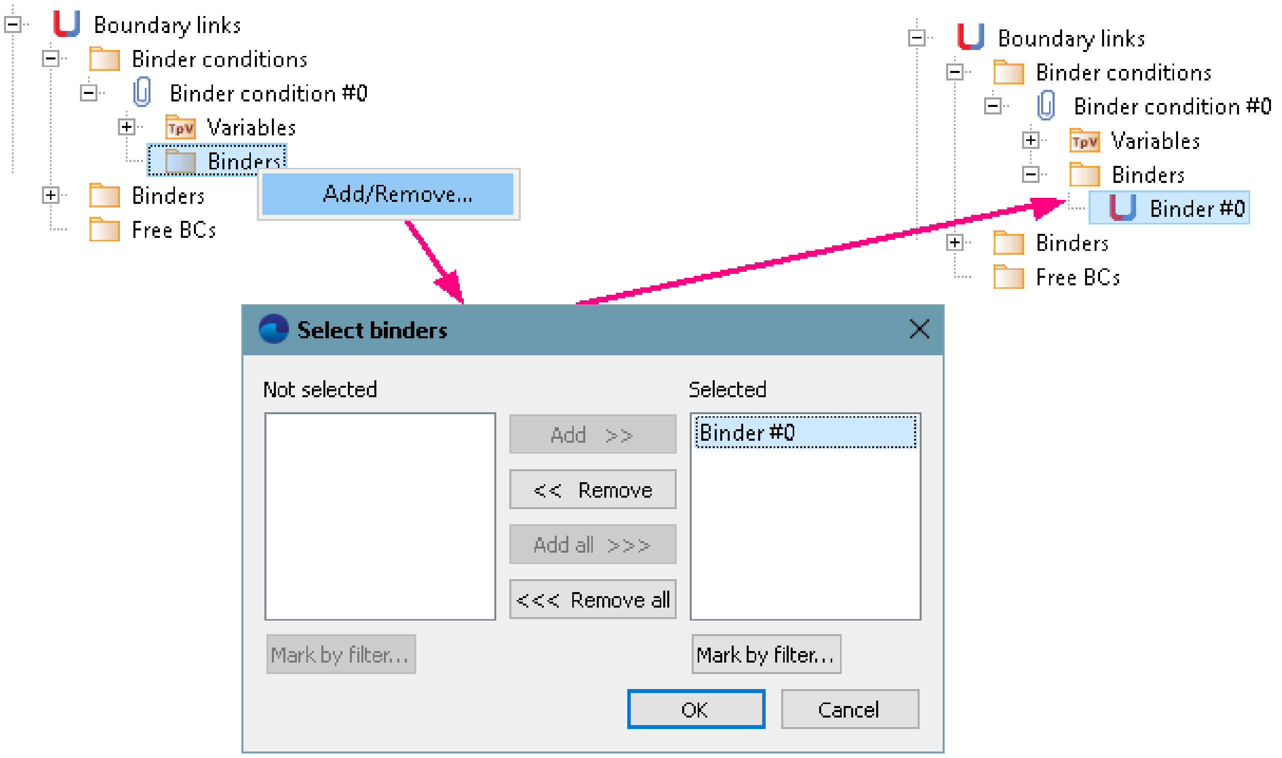
- Select **Create** from the context menu of the folder **Boundary links > Binder conditions**.
- In the **Create binder condition** dialog box, which opens, specify:

Connection type	**= Conjugate temperature**
1st model	**= Steel**
2nd model	**= Air**

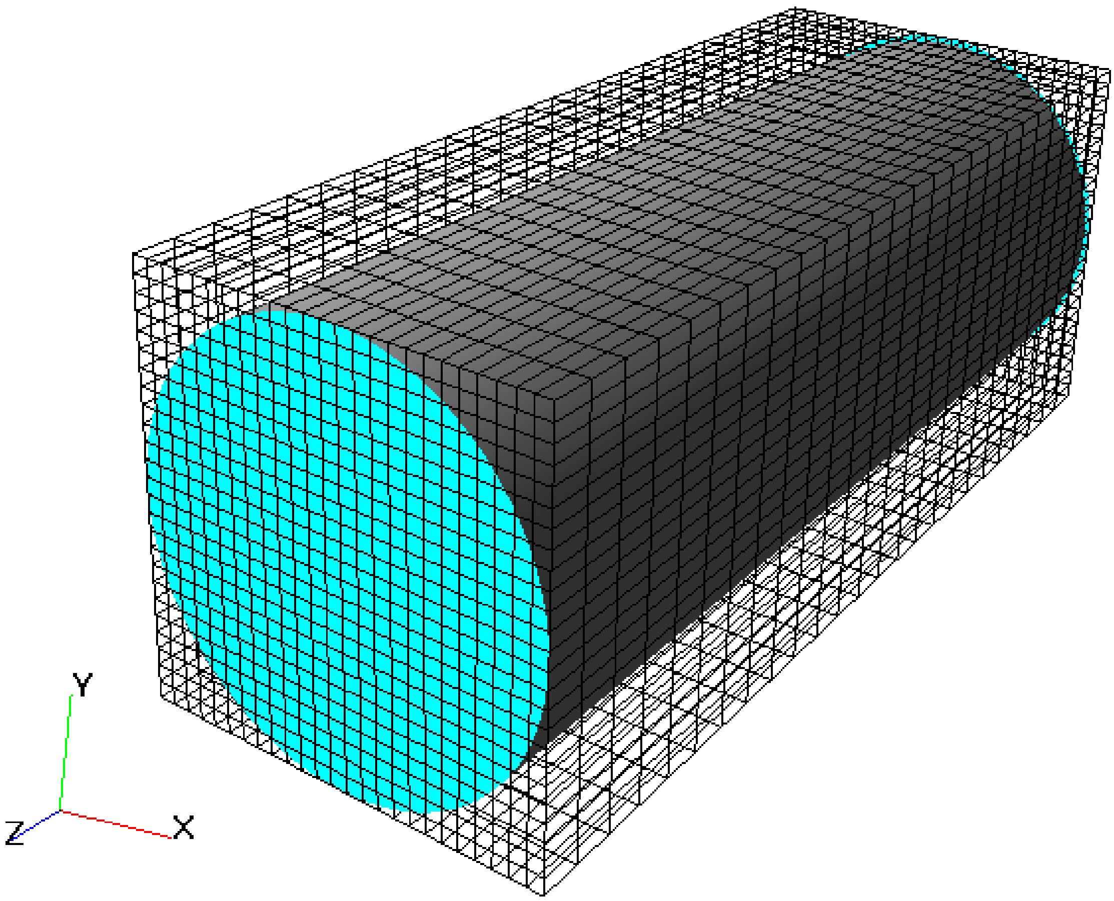


Specify the matching between the **Binder** and the **Binder condition**:

- From the context menu of the folder **Binder condition #0 > Binders**, select **Add/Remove**.
- Select **Binder #0** from the **Not selected** list and click the **Add** button. Then click **OK**.



5.1.1.1.5 Initial grid



Specify in the **Properties** window of the **Initial grid**:

nX	**= 25**
nY	**= 25**
nZ	**= 25**

In the **Properties** window of the **Initial grid** click **Apply**.

5.1.1.1.6 Adaptation of the computational grid

In this example, it is necessary to make an adaptation of the grid within solid walls of the tube and near the wall in the subregion of the flow.

Specify the adaptation of the computational grid within the subregion **Tube**:

- From the context menu of the folder **Computational grid > Adaptation**, select the **Create** command, so **Adaptation #0** will be created.
- From the context menu of the element **Adaptation #0 > Subregions** select the command **Add/Remove** and in the **Select Subregions** dialog box, which opens, place **Tube** into the pane **Selected** and click **OK**.
- From the context menu of the element **Adaptation #0 > Objects** select the command **Add/Remove Objects** and in the **Select objects** dialog box, which opens, place **Computational space** into the pane **Selected** and click **OK**.
- In the **Properties** window of the element **Adaptation #0** specify:

Enabled	**= Yes**
Max level N	**= 1**
Split/Merge	**= Split**
Layers	
Layers for Level N	**= 1**

Specify the adaptation within the subregion **Flow**:

- From the context menu of the folder **Computational grid > Adaptation**, select the **Create** command, so **Adaptation #1** will be created.
- From the context menu of the element **Adaptation #1 > Subregions** select the command **Add/Remove** and in the **Select Subregions** dialog box, which opens, place **Flow** into the pane **Selected** and click **OK**.
- From the context menu of the element **Adaptation #1 > Objects** select the command **Add/Remove Boundary Conditions** and in the **Select boundary conditions** dialog box, which opens, place **Flow : Wall** into the pane **Selected** and click **OK**.
- In the **Properties** window of the element **Adaptation #1** specify:

Enabled	**= Yes**
Max level N	**= 1**
Layers	
Layers for Level N	**= 4**

5.1.1.1.7 Parameters of calculation

Specify in the **Solver** tab in properties of the **Time step** element:

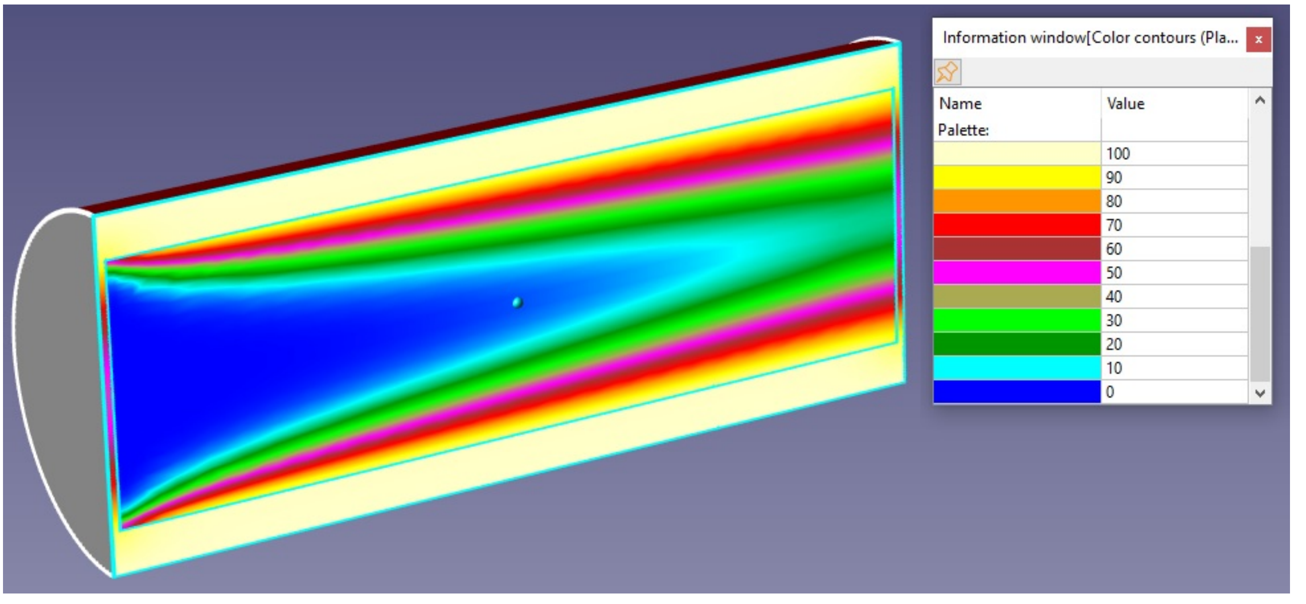
Method	**= In seconds**	
Constant step	**= 0.1**	[s]

5.1.1.1.8 Visualization

To view the dynamics of the solution during the computation, specify visualization of the [temperature distribution](#) in the plane of the flow before the start of computation.

5.1.1.1.8.1 Temperature distribution

Visualization at the step number 50:



- In properties of the layer **Solids** specify:

Clipped	**= Yes**

- In properties of **Plane #0** specify:

Object		
Normal		
	X	**= -1**
Clipping object		**= Yes**

- Create a layer **Color contours** on **Plane #0**.
- In properties of this layer specify:

Variable		
	Variable	**= Temperature**
Value range		
	Mode	**= Manual**
	Max	**= 100**
	Min	**= 0**
Palette		
	Operations	Click (**Load palette from file**) and then select the file `heat.fvpal` (this file locates in the directory where *FlowVision* is installed).

5.1.1.2 Making the project based on several details (an assembly)

Geometry:	`Conj_Convection_TConnect_Part1.STL` `Conj_Convection_TConnect_Part2.STL`
Project:	`Conjugate_Convection_TConnect`

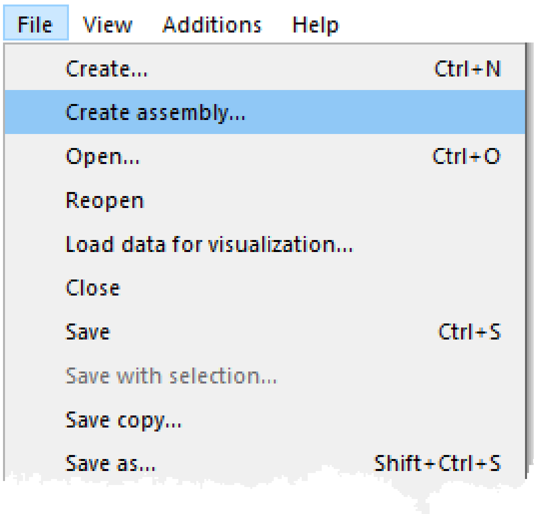
5.1.1.2.1 Computational domain

When a geometric model is prepared based on an *assembly*, the T-shaped surfaces are processed correctly and there is no need of additional modifying the geometric model, as it was in the previous example.

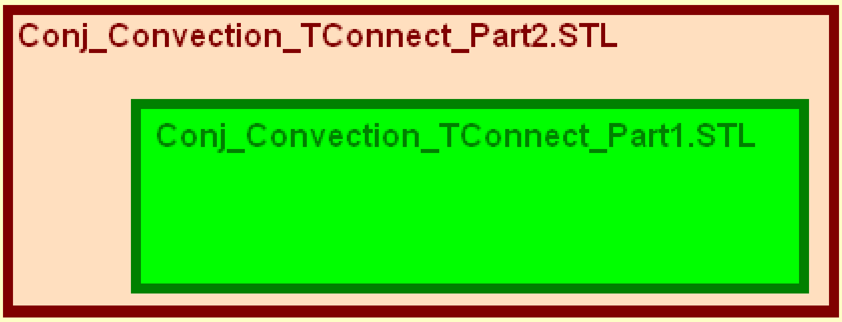
In this case, the geometric model of each subregion must be stored in a separate file. The assembling is done at the step of creating the project when all the files are loaded in *FlowVision*.

To create a project based on an assembly:

- In the **File** menu, select **Create assembly**:

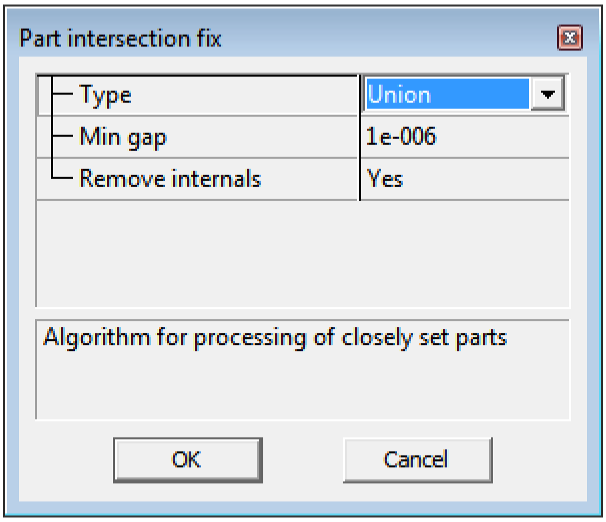


- The **Assembly creation** dialog box will open. Click there the **Add** button and, in a standard operating system's dialog box for file selection, which opens, select the following files:
  - **`Conj_Convection_TConnect_Part1.STL`**
  - **`Conj_Convection_TConnect_Part2.STL`**
- The selected files will appear in the **Files in assembly** pane in the **Assembly creation** dialog box.

> The files are to be displayed in the **Files in assembly** pane of the **Assembly creation** dialog box in the following order:
>
> 1) **`Conj_Convection_TConnect_Part2.STL`**
> 2) **`Conj_Convection_TConnect_Part1.STL`**
>
> This order is important, because *the external subregion must be specified on the first position* in the assembly list, and *internal subregions must be specified on other places*.
>
> 
>

If necessary, place the file **`Conj_Convection_TConnect_Part2.STL`** on the first position in the list applying buttons **Move Up** and/or **Move Down**.

- Click **OK** in the **Assembly creation** dialog box.
- The **Part intersection fix** dialog box will open. Leave there the default settings and click **OK**.



- Specify the name of the project.

5.1.1.2.2 Physical model

In the folder **Substances**:

- Create **Substance #0**.
- Specify the following properties of **Substance #0**:

Name	**= Steel**	
Aggregative state	**= Solid**	
Molar mass		
Value	**= 0.056**	[kg mole^{-1}]
Density		
Value	**= 7900**	[kg m^{-3}]
Thermal conductivity		
Value	**= 45**	[W m^{-1}K^{-1}]
Specific heat		
Value	**= 457**	[J kg^{-1} K^{-1}]

- Create **Substance #0**.
- In properties of **Substance #0** specify:

Name	**= Air**	
Aggregative state	**= Gas**	
Molar mass		
Value	**= 0.0289**	[kg mole^{-1}]
Density		
Value	**= 1**	[kg m^{-3}]
Viscosity		
Value	**= 1.82e-5** $^{*)}$	[kg m^{-1}s^{-1}]
Thermal conductivity		
Value	**= 0.026**	[W m^{-1}K^{-1}]
Specific heat		
Value	**= 1009**	[J kg^{-1} K^{-1}]

*) **1.82e-5** is notation for 1.82×10^{-5}.

In the folder **Phases**:

- Create a continuous **Phase #0**.
- In properties of **Phase #0**, specify:

Name = **Steel**

- Load the **Steel** substance into the **Substances** folder of this **Phase**.
- In properties of the folder **Steel > Physical processes** specify:

Heat transfer = **Heat transfer via h**

- Create a continuous **Phase #0**.
- In properties of **Phase #0**, specify:

Name = **Air**

- Load the **Air** substance into the **Substances** folder of this **Phase**.
- In properties of the folder **Air > Physical processes** specify:

Heat transfer = **Heat transfer via h**
Motion = **Navier-Stokes model**

In the folder **Models**:

- Create **Model #0**.
- In properties of **Model #0**, specify:

Name = **Steel**

- Add phase **Steel** into subfolder **Steel > Phases**.
- Create **Model #0**.
- In properties of **Model #0**, specify:

Name = **Air**

- Add phase **Air** into subfolder **Air > Phases**.
- In **Init. data #0** specify:

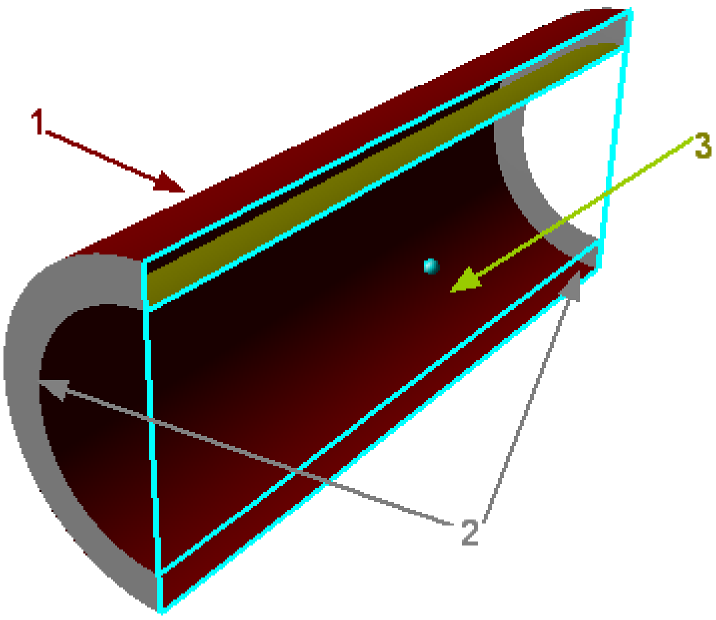
Velocity (Air)

X	**= 0.1**	[m s^{-1}]
Y	**= 0**	[m s^{-1}]
Z	**= 0**	[m s^{-1}]

5.1.1.2.3 Boundary conditions

In the **Properties** window of the **Subregion** between the external and the internal surfaces, specify:

Name = **Tube**
Model = **Steel**



Specify the following boundary conditions (on the inner side of the outer surface and the outer side of the inner surface):

Boundary 1

Name	**= Outer wall**	
Type	**= Wall**	
Color	**= Maroon**	
Variables		
Temperature(Steel)	**= Temperature**	
Value	**= 100**	[K]

Boundary 2

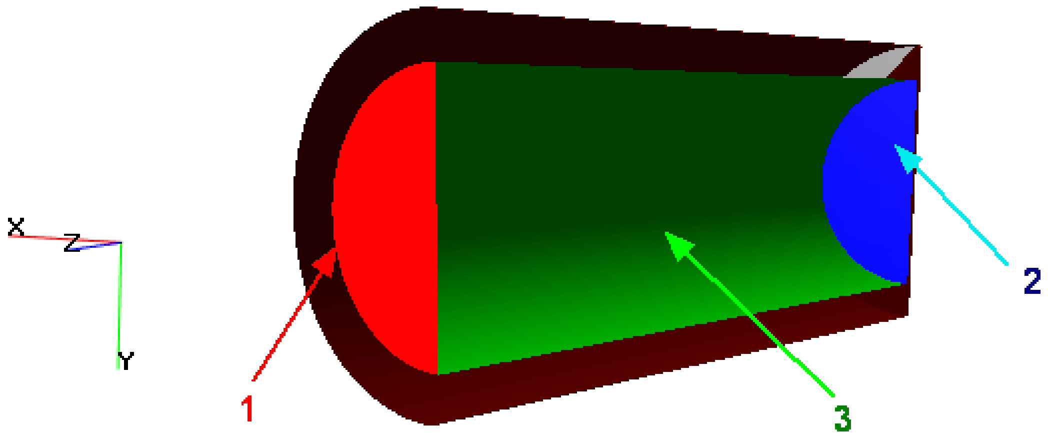
Name	**= Wall**
Type	**= Wall**
Color	**= Gray**
Variables	
Temperature(Steel)	**= Zero gradient**

Boundary 3

Name	**= Inner wall**
Type	**= Connected**
Color	= **Yellow**

In the **Properties** window of the **Subregion** within the inner surface, specify:

Name	**= Flow**
Model	**= Air**



Specify the following boundary conditions (on the inner side of the inner surface):

Boundary 1

Name	**= Inlet**	
Type	**= Inlet/Outlet**	
Color	**= Red**	
Variables		
Temperature(Air)	**= Temperature**	
Value	**= 0**	[K]
Velocity(Air)	**= Normal mass velocity**	
Mass velocity	**= 0.1**	[kg $(m^2s)^{-1}$]

Boundary 2

Name	**= Outlet**	
Type	**= Free Outlet**	
Color	**= Blue**	
Variables		
Temperature(air)	**= Zero gradient**	
Velocity(Air)	**= Pressure**	
Value	**= 0**	[Pa]

Boundary 3

Name	**= Wall**
Type	**= Connected**
Color	**= Green**

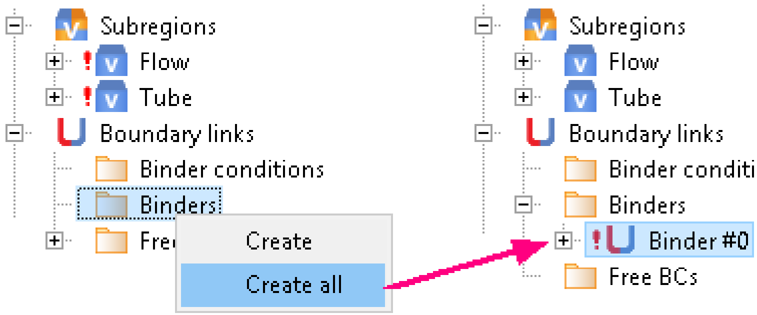
5.1.1.2.4 Binding the subregions

Binding the boundary conditions consists of the following steps:

- creating a **Binder** of two **Boundary conditions**, which types are **Connected**
- creating a **Binder condition**
- matching the **Binder** and the **Binder condition**

Create **Binder #0**:

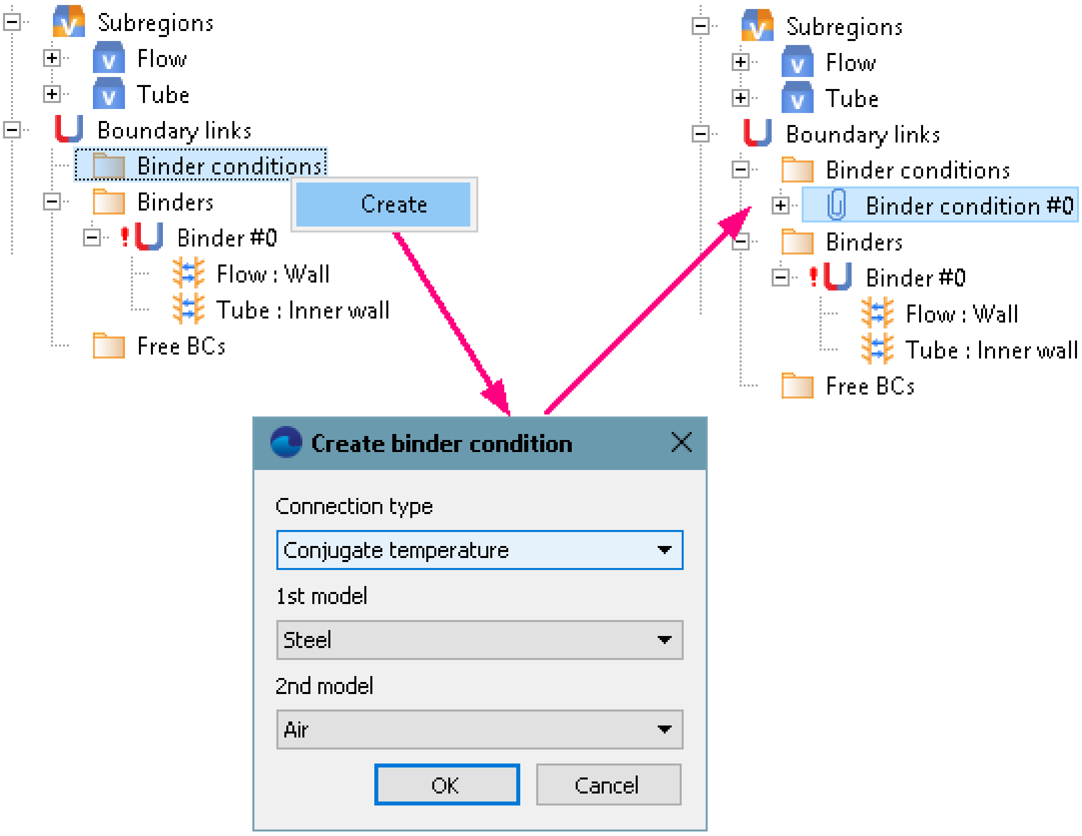
- Select **Create all** from the context menu of the folder **Boundary links > Binders**:



Create **Binder condition #0**:

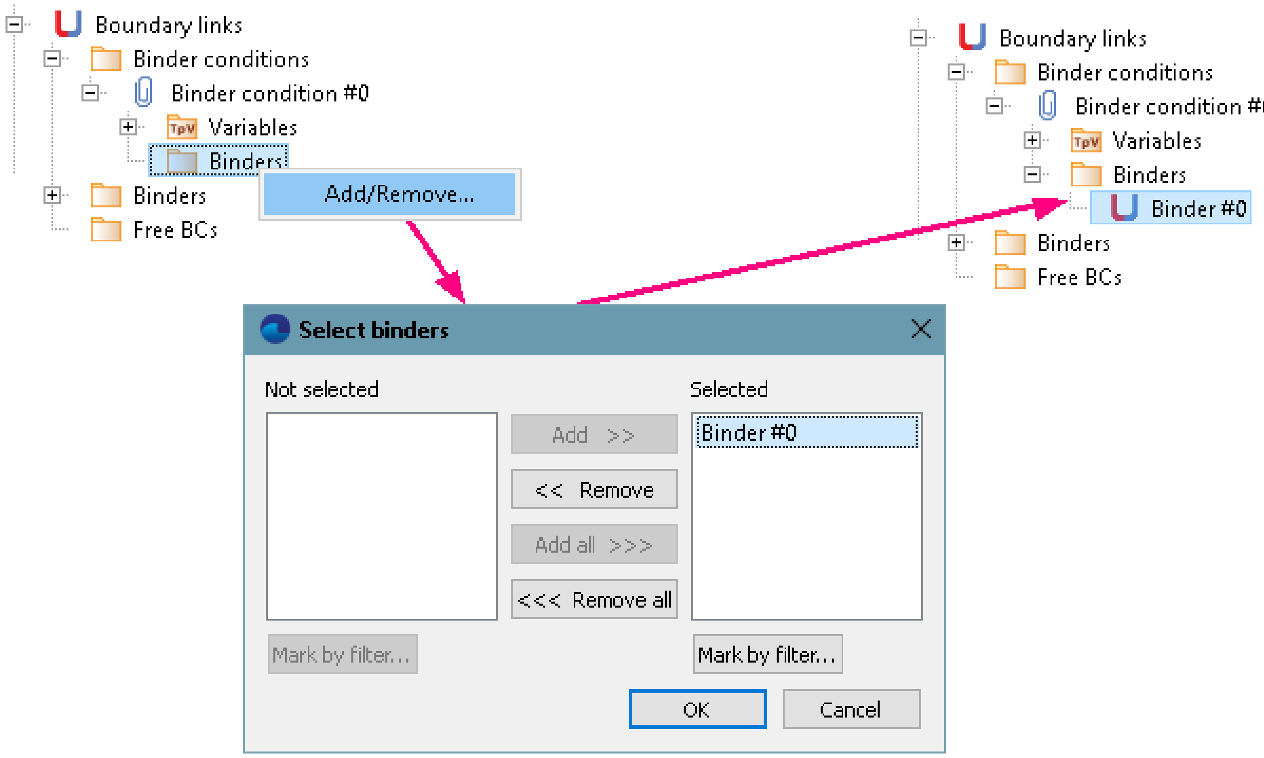
- Select **Create** from the context menu of the folder **Boundary links > Binder conditions**.
- In the **Create binder condition** dialog box, which opens, specify:

Connection type	**= Conjugate temperature**
1st model	**= Steel**
2nd model	**= Air**

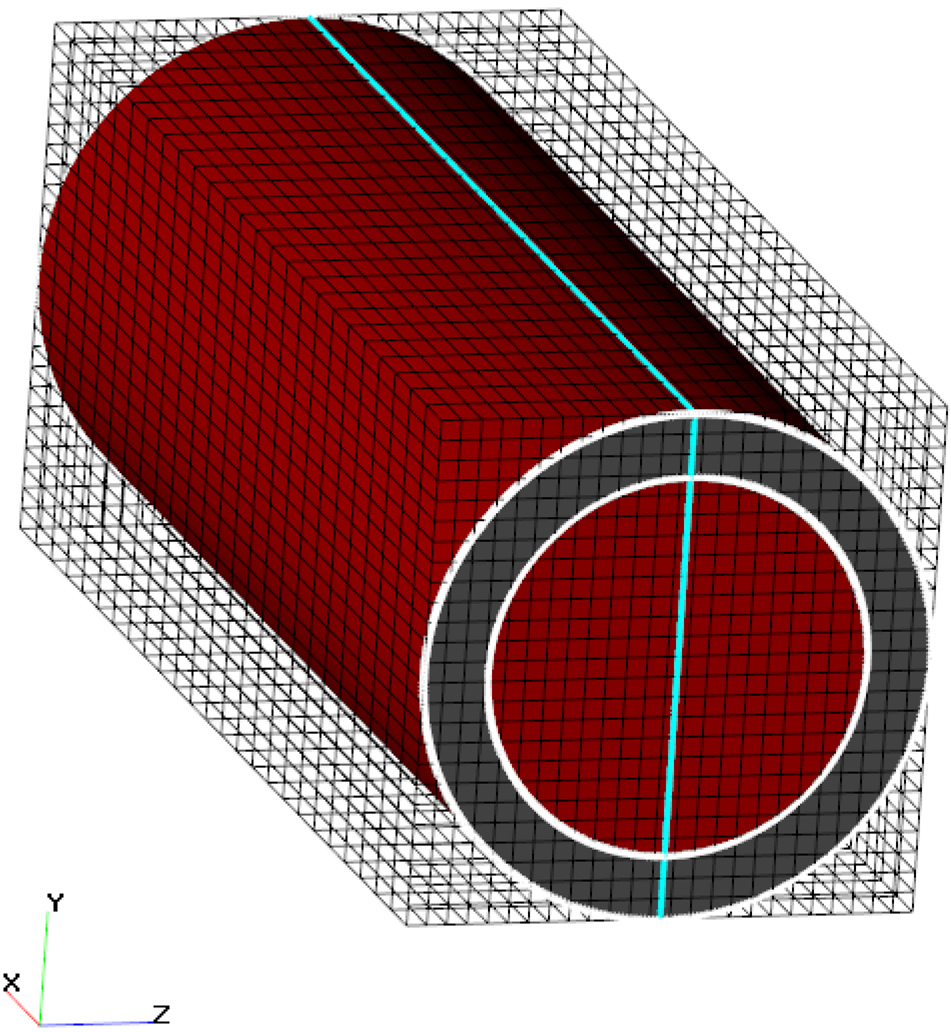


Specify the matching between the **Binder** and the **Binder condition**:

- From the context menu of the folder **Binder condition #0 > Binders**, select **Add/Remove**.
- Select **Binder #0** from the **Not selected** list and click the **Add** button. Then click **OK**.



5.1.1.2.5 Initial grid



Specify in the **Properties** window of the **Initial grid**:

nX	**= 25**
nY	**= 25**
nZ	**= 25**

In the **Properties** window of the **Initial grid** click **Apply**.

5.1.1.2.6 Adaptation of the computational grid

In this example, it is necessary to make an adaptation of the grid within solid walls of the tube and near the wall in the subregion of the flow.

Specify the adaptation of the computational grid within the subregion **Tube**:

- From the context menu of the folder **Computational grid > Adaptation**, select the **Create** command, so **Adaptation #0** will be created.
- From the context menu of the element **Adaptation #0 > Subregions** select the command **Add/Remove** and in the **Select Subregions** dialog box, which opens, place **Tube** into the pane **Selected** and click **OK**.
- From the context menu of the element **Adaptation #0 > Objects** select the command **Add/Remove Objects** and in the **Select objects** dialog box, which opens, place **Computational space** into the pane **Selected** and click **OK**.
- In the **Properties** window of the element **Adaptation #0** specify:

Enabled	**= Yes**
Max level N	**= 1**
Split/Merge	**= Split**
Layers	
Layers for Level N	**= 1**

Specify the adaptation within the subregion **Flow**:

- From the context menu of the folder **Computational grid > Adaptation**, select the **Create** command, so **Adaptation #1** will be created.
- From the context menu of the element **Adaptation #1 > Subregions** select the command **Add/Remove** and in the **Select Subregions** dialog box, which opens, place **Flow** into the pane **Selected** and click **OK**.
- From the context menu of the element **Adaptation #1 > Objects** select the command **Add/Remove Boundary Conditions** and in the **Select boundary conditions** dialog box, which opens, place **Flow: Wall** into the pane **Selected** and click **OK**.
- In the **Properties** window of the element **Adaptation #1** specify:

Enabled	**= Yes**
Max level N	**= 1**
Layers	
Layers for Level N	**= 4**

5.1.1.2.7 Parameters of calculation

Specify in the **Solver** tab in properties of the **Time step** element:

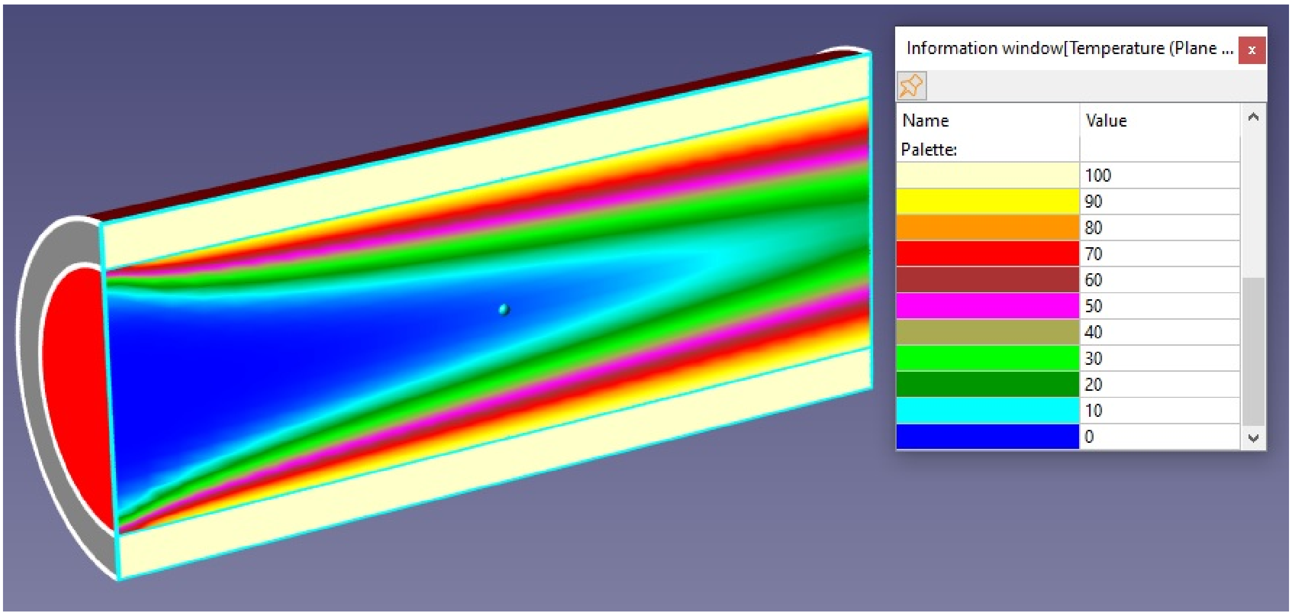
Method	**= In seconds**	
Constant step	**= 0.1**	[s]

5.1.1.2.8 Visualization

To view the dynamics of the solution during the computation, specify visualization of the [temperature distribution](#) in the plane of the flow before the start of computation.

5.1.1.2.8.1 Temperature distribution

Visualization at the step number 50:



- In the **Properties** window of the layer **Solids** specify:

Clipped = **Yes**

- In the **Properties** window of **Plane #0** specify:

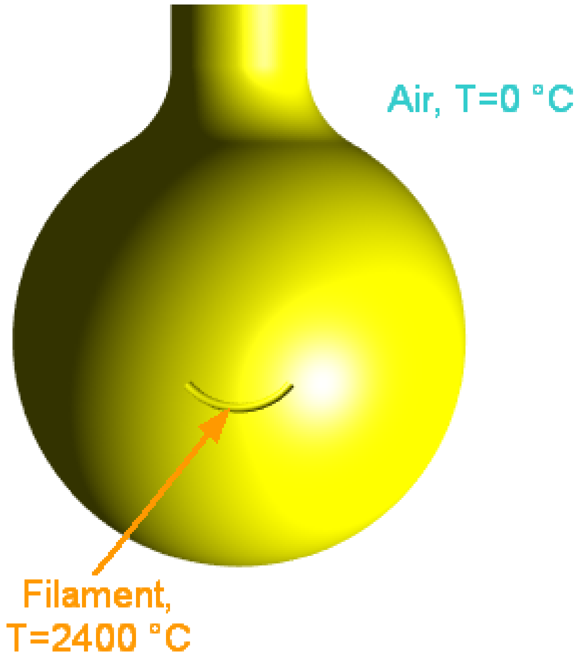
Object
Normal
X = **0**
Y = **0**
Z = **-1**
Clipping object = **Yes**

- Create a layer **Color contours** on **Plane #0**.
- In the **Properties** window of the layer specify:

Variable
Variable = **Temperature**
Value range
Mode = **Manual**
Max = **100**
Min = **0**
Palette
Operations Click (**Load palette from file**) and then select the file **`heat.fvpal`** (this file locates in the directory where *FlowVision* is installed).

5.1.2 Conjugate radiation heat transfer

Consider the natural convection of air around the lamp bulb, heated from the inside by a filament.



Parameters of the problem setting

Boundary parameters:

The temperature of the filament	T_s	=2400	[K]
Ambient temperature	T_{air}	= 0	[K]
The emissivity of the bulb		= 10^{-4}	

Properties of the air:

Molar mass	M	= 0.0289	[kg mole^{-1}]
Viscosity	μ	=1.82×10^{-5}	[kg m^{-1}s^{-1}]
Thermal conductivity	λ	= 0.026	[W m^{-1}K^{-1}]
Specific heat	c_p	= 1009	[J kg^{-1} K^{-1}]

Geometry: `Lamp.STL`

Project: `Lamp`

5.1.2.1 Computational domain

In *FlowVision*, the surface from which a **Subregion** is formed, can be defined after the creation of the project. The surface geometry is imported from a file or is created based on objects of **Preprocessor**. In this project, you have to create an outer surface of the **Subregion**, where motion of the air is simulated.

Create an outer surface of the **Subregion**:

- In the folder **Objects** on the **Preprocessor** tab, create a **Box**.
- In the **Properties** window of **Box #0**, specify:

Size

X	**= 0.5**	[m]
Y	**= 0.5**	[m]
Z	**= 0.5**	[m]

- From the context menu of **Box #0**, select **Build into the main geometry.**
- In the **Properties** window of the new just created **Subregion**, specify:

Name **= External environment**

In the **Properties** window of the **Subregion**, which corresponds to the lamp, specify:

Name **= Lamp**

5.1.2.2 Physical model

In the **Properties** window of the element **General settings** specify:

Gravity vector		
X	**= 0**	[m s^{-2}]
Y	**=9.8**	[m s^{-2}]
Z	**=0**	[m s^{-2}]
g-Density	**= 1.286**	[kg m^{-3}]

In the folder **Substances**:

- Create **Substance #0.**
- Specify the following properties of **Substance #0**:

Name	**= Air**	
Aggregative state	**= Gas**	
Properties		
Molar mass	**= Constant**	
Value	**= 0.0289**	[kg $mole^{-1}$]
Density	**= Ideal gas law**	
Viscosity	**=Constant**	
Value	**=1.82e-5**	[kg $m^{-1}s^{-1}$]
Thermal conductivity	**=Constant**	
Value	**= 0.026**	[W $m^{-1}K^{-1}$]
Specific heat	**=Constant**	
Value	**= 1009**	[J kg^{-1} K^{-1}]

- Create **Substance #0.**
- Specify the following properties of **Substance #0**:

Name	**= Vacuum**	
Aggregative state	**= Solid**	
Properties		
Density	=**Constant**	
Value	**= 0.001**	[kg m^{-3}]
Thermal conductivity	**=Constant**	
Value	= **1e-8**	[W $m^{-1}K^{-1}$]
Specific heat	=**Constant**	
Value	= **1e+8**	[J kg^{-1} K^{-1}]

In the folder **Phases**:

- Create a continuous **Phase #0.**

- In the **Properties** window of **Phase #0**, specify:

  Name = Air

- Add the substance **Air** into the folder **Substances** of the phase **Air**.
- In the **Properties** window of the folder **Phases > Air > Physical processes**, specify:

  Motion = Navier-Stokes model

  Heat transfer = Heat transfer via h

- Create a continuous **Phase #0**.
- In the **Properties** window of **Phase #0**, specify:

  Name = Vacuum

- Add the substance **Vacuum** into the folder **Substances** of the phase **Vacuum**.
- In the **Properties** window of the folder **Phases > Vacuum > Physical processes**, specify:

  Heat transfer = Heat transfer via h

  Radiation = P1

In the folder **Models**:

- Create **Model #0**.
- In the **Properties** window of **Model #0**, specify:

  Name = Air

- Add phase **Air** into subfolder **Air > Phases**.
- Create **Model #0**.
- In the **Properties** window of **Model #0**, specify:

  Name = Vacuum

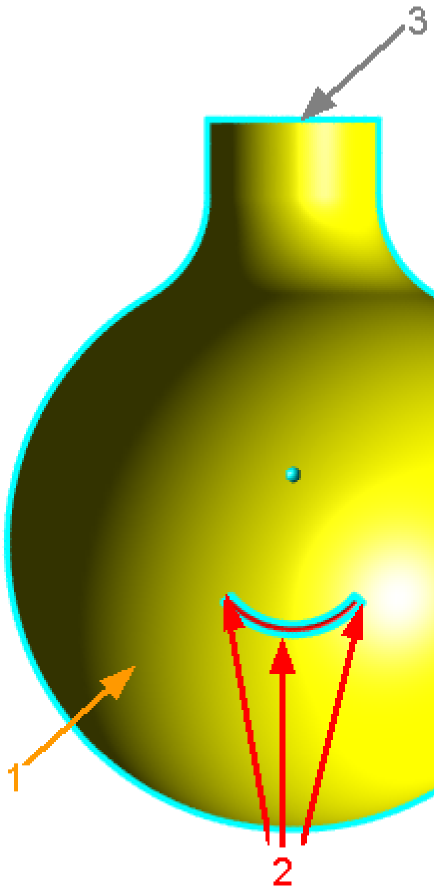
- Add phase **Vacuum** into subfolder **Vacuum > Phases**.

5.1.2.3 Specifying boundary conditions (Part 1)

> In this exercise some parameters of connected boundary conditions can be only set after binding the subregions.
>
> After binding the subregions, it is necessary to specify some more parameters of the boundary conditions.

In the **Properties** window of the subregion **Lamp**, specify:

Model = Vacuum



Specify the following boundary conditions:

Boundary 1

Name	= **Glass bulb**	
Type	= **Connected**	

Boundary 2

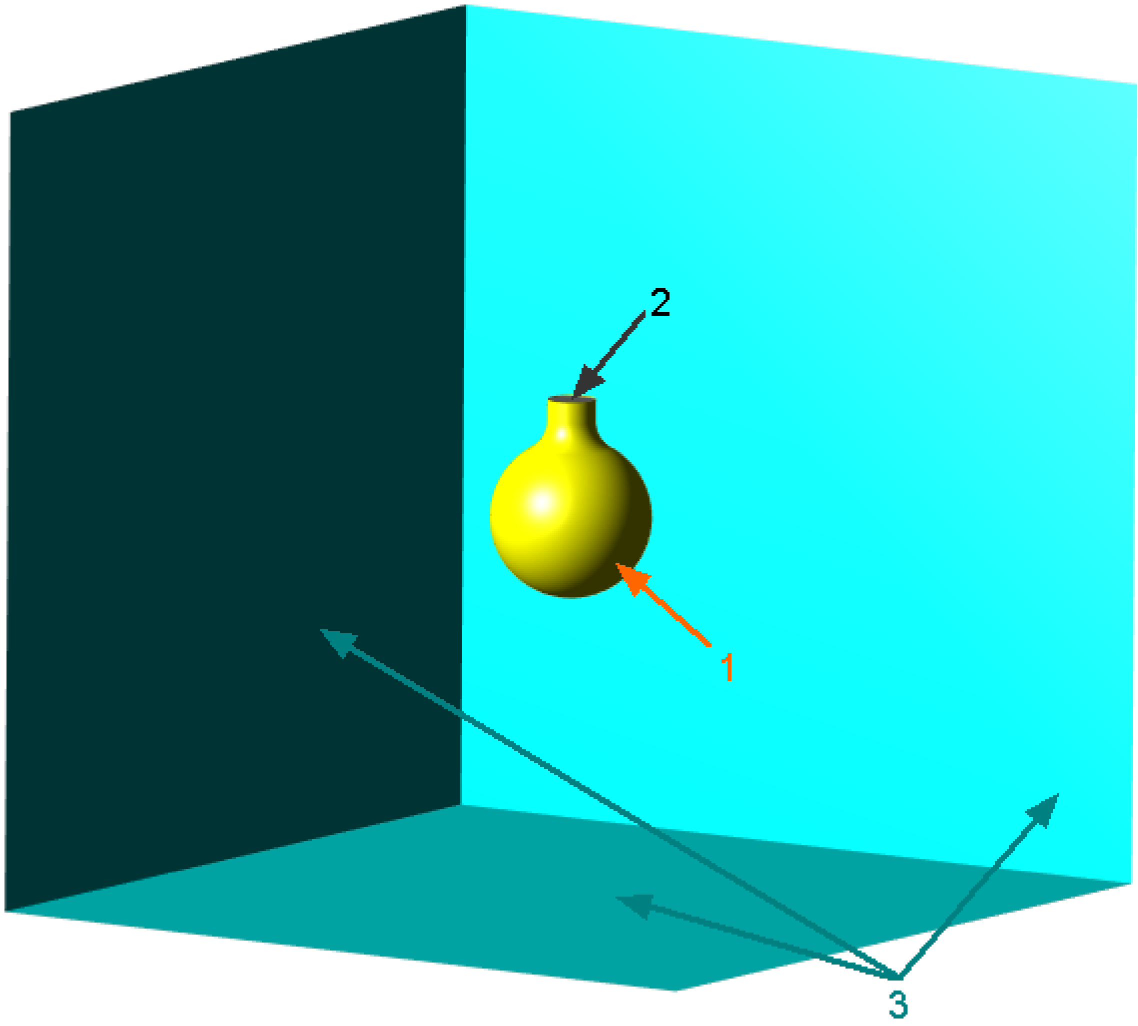
Name	= **Filament**	
Type	= **Wall**	
Variables		
Temperature (Vacuum)	= **Temperature**	
Value	= **2400**	[K]
Radiation density (Vacuum)	= **Calculating of radiation flux density**	
Blackness	= **1**	

Boundary 3

Name	= **Wall**	
Type	= **Wall**	
Variables		
Temperature (Vacuum)	= **Zero gradient**	
Radiation density (Vacuum)	= **Calculating of radiation flux density**	
Blackness	= **1**	

In the **Properties** window of the subregion **External environment**, specify:

Model = **Air**



Specify the following boundary conditions:

Boundary 1

Name = **Glass bulb**

Type = **Connected**

Boundary 2

Name = **Wall**

Type = **Wall**

Variables

Temperature(Air) = **Zero gradient**

Velocity(Air) = **No slip**

Boundary 3

Name = **Outlet**

Type = **Inlet/Outlet**

Variables

Temperature(Air) = **Temperature**

Value	**= 0**	[K]
Velocity (Air)	**= Total pressure**	
Value	**= 0**	[Pa]

5.1.2.4 Binding the subregions

Create **Binder #0**:

- Select **Create** from the context menu of the folder **Binders**.
- In the **Create binder** window, which opens, specify:

Free BC list	**= Lamp: Glass bulb**
Free BC list	**= External environment: Glass bulb**

Create **Binder condition #0**:

- Select **Create** from the context menu of the folder **Binder conditions**.
- In the **Create binder condition** dialog box, which opens, specify:

Connection type	**= Conjugate temperature**
1st model	**= Vacuum**
2nd model	**= Air**

Specify the matching between the **Binder** and the **Binder condition**:

- From the context menu of the folder **Binder condition #0 > Binders**, select **Add/Remove**.
- Add **Binder #0**.

5.1.2.5 Specifying boundary conditions (Part 2)

For the subregion **Lamp**, specify the remaining boundary conditions' parameters (you add them to the parameters, which have already been specified):

In the **Properties** window of the boundary condition **Glass bulb**, specify:

Variables

Radiation density (Vacuum)	= **Calculating of radiation flux density**
Blackness	**= 0.0001**

5.1.2.6 Initial grid

Specify in the **Properties** window of the **Initial grid**:

nX	**= 5**
nY	**= 5**
nZ	**= 5**

In the **Properties** window of the **Initial grid** click **Apply**.

5.1.2.7 Adaptation

Specify an **Adaptation** in the subregion **Lamp**.

- From the context menu of the folder **Computational grid > Adaptation**, select the **Create** command, so **Adaptation #0** will be created.
- From the context menu of the element **Adaptation #0 > Subregions** select the command **Add/Remove** and in the **Select Subregions** dialog box, which opens, place **Lamp** into the pane **Selected** and click **OK**.

- From the context menu of the element **Adaptation #0 > Objects** select the command **Add/Remove Objects** and in the **Select objects** dialog box, which opens, place **Computational space** into the pane **Selected** and click **OK**.
- In the **Properties** window of the element **Adaptation #0** specify:

Enabled	**= Yes**
Max level N	**= 4**
Split/Merge	**= Split**
Layers	
Layers for Level N	**= 4**
Layers for Level N-1	**= 4**
Layers for Level N-2	**= 4**
Layers for Level N-3	**= 4**

Specify two **Adaptations** in the subregion **External environment**:

Adaptation #1 will be created in the volume within a geometry object (cylinder), and **Adaptation #2** will be created on the surface of the glass bulb of the lamp:

- In **Preprocessor**, in the folder **Objects**, create a **Cone/cylinder** object.
- In the **Properties** window of the object **Cone/cylinder #0**, specify:

Location		
Reference point		
X	**= 0**	[m]
Y	**= -0.26**	[m]
Z	**= 0**	[m]
Axis X		
X	**= 0**	
Y	**= 1**	
Z	**= 0**	
Parameters		
Height	**= 0.2**	[m]
Radius 1	**= 0.025**	[m]
Radius 2	**= 0.025**	[m]
Base ratio	**= 1**	

- From the context menu of the folder **Computational grid > Adaptation**, select the **Create** command, so **Adaptation #1** will be created.
- From the context menu of the element **Adaptation #1 > Subregions** select the command **Add/Remove** and in the **Select Subregions** dialog box, which opens, place **External environment** into the pane **Selected** and click **OK**.
- From the context menu of the element **Adaptation #1 > Objects** select the command **Add/Remove Objects** and in the **Select objects** dialog box, which opens, place **Cone/cylinder #0** into the pane **Selected** and click **OK**.
- In the **Properties** window of the element **Adaptation #1** specify:

Enabled	**= Yes**
Max level N	**= 4**
Split/Merge	**= Split**
Layers	
Layers for Level N	**= 4**
Layers for Level N-1	**= 4**

Layers for Level N-2	**= 4**
Layers for Level N-3	**= 4**

- From the context menu of the folder **Computational grid > Adaptation**, select the **Create** command, so **Adaptation #2** will be created.
- From the context menu of the element **Adaptation #2 > Subregions** select the command **Add/Remove** and in the **Select Subregions** dialog box, which opens, place **External environment** into the pane **Selected** and click **OK**.
- From the context menu of the element **Adaptation #2 > Objects** select the command **Add/Remove Boundary Conditions** and in the **Select boundary conditions** dialog box, which opens, place **External environment : Glass bulb** into the pane **Selected** and click **OK**.
- In the **Properties** window of the element **Adaptation #2** specify:

Enabled		**= Yes**
Max level N		**= 5**
Split/Merge		**= Split**
Layers		
	Layers for Level N	**= 6**
	Layers for Level N-1	**= 5**
	Layers for Level N-2	**= 4**
	Layers for Level N-3	**= 5**
	Layers for Level N-4	**= 6**

5.1.2.8 Parameters of calculation

Specify in the **Solver** tab in properties of the **Time step** element:

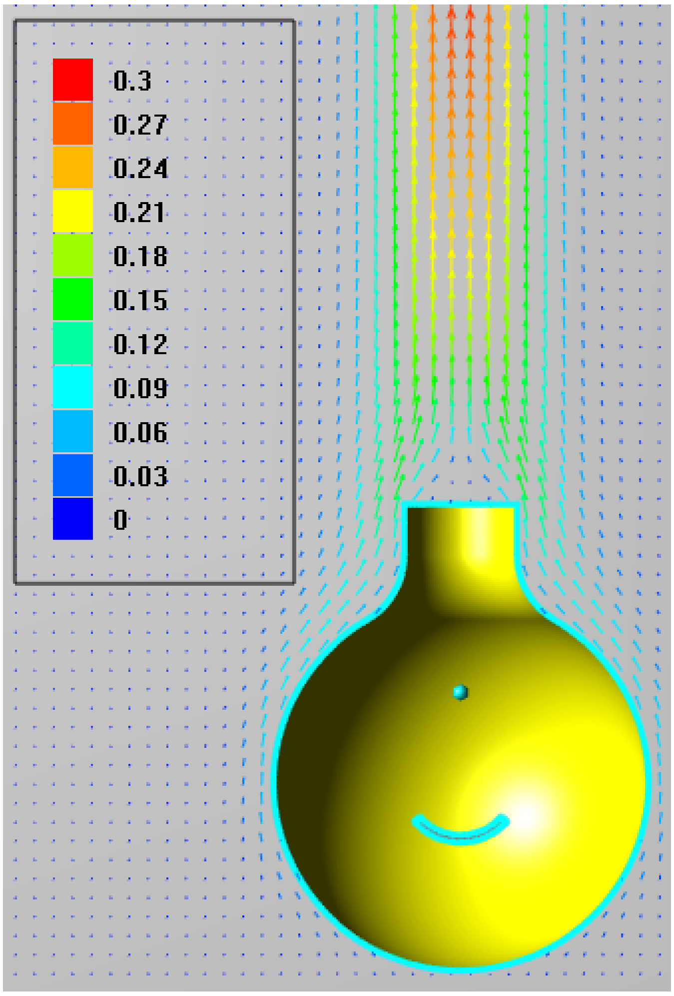
Method	**= Via CFL number**	
Convective CFL	**= 100**	
Max step	**= 1**	[s]

5.1.2.9 Visualization

To view the dynamics of the solution during the computation, prior to computation, specify visualization of:

1. Velocity distribution in the plane of the flow
2. Temperature distribution in the plane of the flow

5.1.2.9.1 Velocity distribution



- Specify orientation of **Plane #0** (its object in the project tree is **Objects > Plane #0** in the **Postprocessor** tab). Specify in its **Properties** window:

  Object

  Normal

  X = 0

  Y = 0

  Z = 1

  (to direct a **Plane**'s normal along the axis **Z**, you can also click the **Operations >** Z↓ button in the **Properties** window of the **Plane**)

- Create a layer **Vectors** on **Plane #0**.
- In the **Properties** window of the **Vectors** specify:

  Grid

  Size 1 = 100

  Size 2 = 100

  Coloring

  Variable

  Variable = Velocity

  Value range

  Mode = Manual

  Max = 0.3

  Min = 0

Palette

Appearance

Enabled = **Yes**

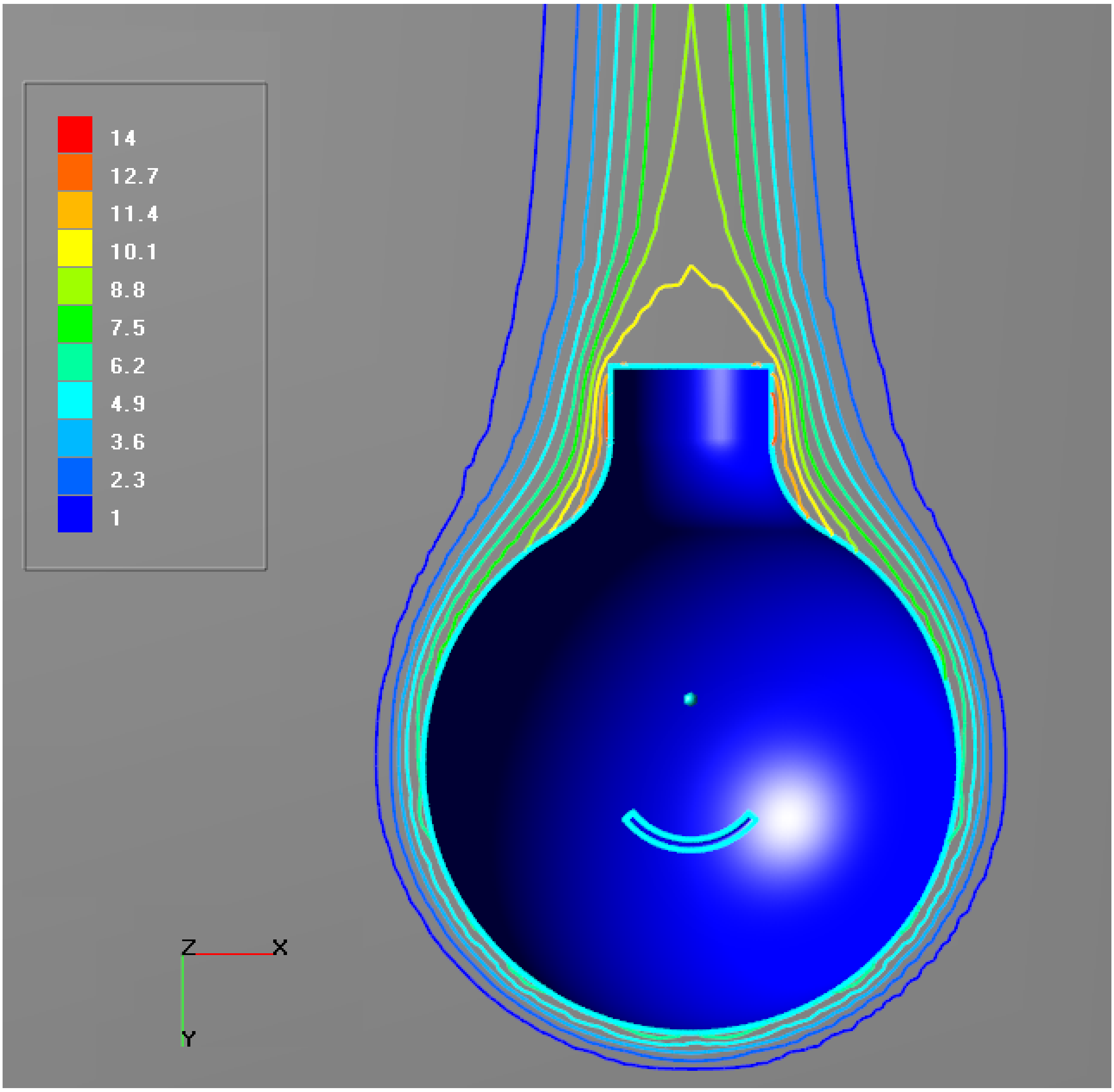
Color = **Black**

The program will automatically specify the variable, which is used to build the vectors, **Variable > Variable = Velocity**.

Note:

If you wish to hide some layers or objects displayed in the **View** window, use the **Hide** command from their context menus.

5.1.2.9.2 Temperature distribution



- Create a layer **Color contours** on **Plane #0**.
- In the **Properties** window of the **Color contours** specify:

Variable

Variable = **Temperature**

Value range

Mode = **Manual**

Max = **14**

Min = **1**

Method = **Isolines**

Palette

Appearance

Enabled = **Yes**

Note:

If you wish to hide some layers or objects displayed in the **View** window, use the **Hide** command from their context menus.

5.2 Rotation

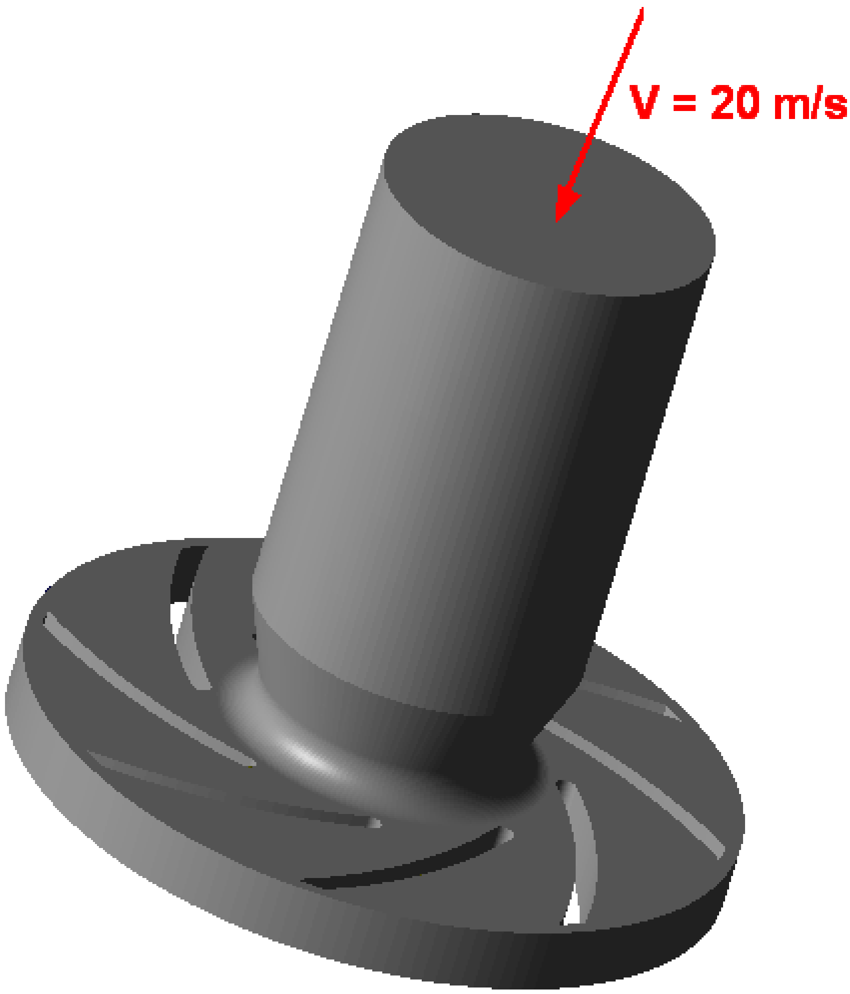
FlowVision can take into account the rotation of surfaces in the selected coordinate system. The program can take into account both normal and tangential components of the rotational speed or only the tangential component.

To specify a rotation, you have to:

- Create a **Local coordinate system** and **Rotation**.
- Specify a rotation on the boundary condition, which is specified mounted on the rotating surface.
- If necessary, specify a **Rotation** on the **Region**.

5.2.1 Rotor

This example illustrates simulation of turbulent viscous motion of incompressible liquid between the blades of a rotating rotor.



Parameters of the problem setting

Dimensions:

Length of the inlet passage	L	= 0.17	[m]
Radius of the rotor	R	= 0.1	[m]

Inflow parameters:

Velocity on inlet:	V_{inl}	= 20	[m s^{-1}]
Speed of rotation	ω	= 300	[radian s^{-1}]

Parameters of the substance:

Density	ρ	= 1	[kg m^{-3}]
Viscosity	μ	= 1.82 * 10^{-5}	[kg m^{-1}s^{-1}]

Reynolds number: $$\mathrm{Re} = \frac{V_{inl} D\rho}{\mu} = \frac{20 \cdot 0.05 \cdot 1}{1.82 \cdot 10^{-5}} \approx 5*10^4$$

Geometry: `Rotor.STL`

Project: `Rotor`

5.2.1.1 Physical model

In the folder **Substances**:

- Create **Substance #0**.
- Specify the following properties of **Substance #0**:

Aggregative state	**= Gas**	
Molar mass		
Value	**= 0.0289**	[kg mole^{-1}]
Density		
Value	**= 1**	[kg m^{-3}]
Viscosity		
Value	**= 1.82e-5**	[kg m^{-1}s^{-1}]
Thermal conductivity		
Value	**= 0.026**	[W m^{-1}K^{-1}]
Specific heat		
Value	**= 1009**	[J kg^{-1} K^{-1}]

In the folder **Phases**:

- Create a continuous **Phase #0**.
- In **Phase #0** add **Substance #0** into the folder **Substances**.
- Specify properties of the folder **Phase #0 > Physical processes**:

Motion	**= Navier-Stokes model**
Turbulence	**= KES**

In the folder **Models**:

- Create **Model #0**.
- Add **Phase #0** into subfolder **Model #0 > Phases**.
- In properties of the element **Init. data #0 > Pulsations (Phase #0)** specify **Value = 0.01**.
- In properties of the element **Init. data #0 > Turbulent scale (Phase #0)** specify **Value = 0.01** [m].

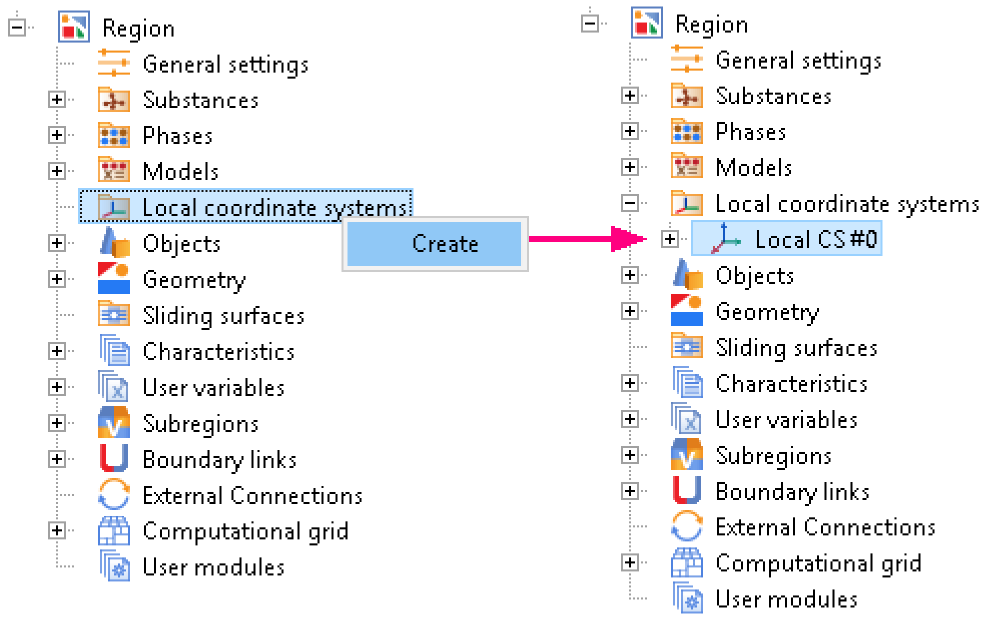
5.2.1.2 Rotation

To specify a rotation of a surface, it is necessary to follow the next steps:

- Create a **Local coordinate system** relative to which the rotation will occur.
- Specify a **Rotation** in this **Local coordinate systems**.
- Specify the **Rotation** on a surface.

Create a **Local coordinate system**:

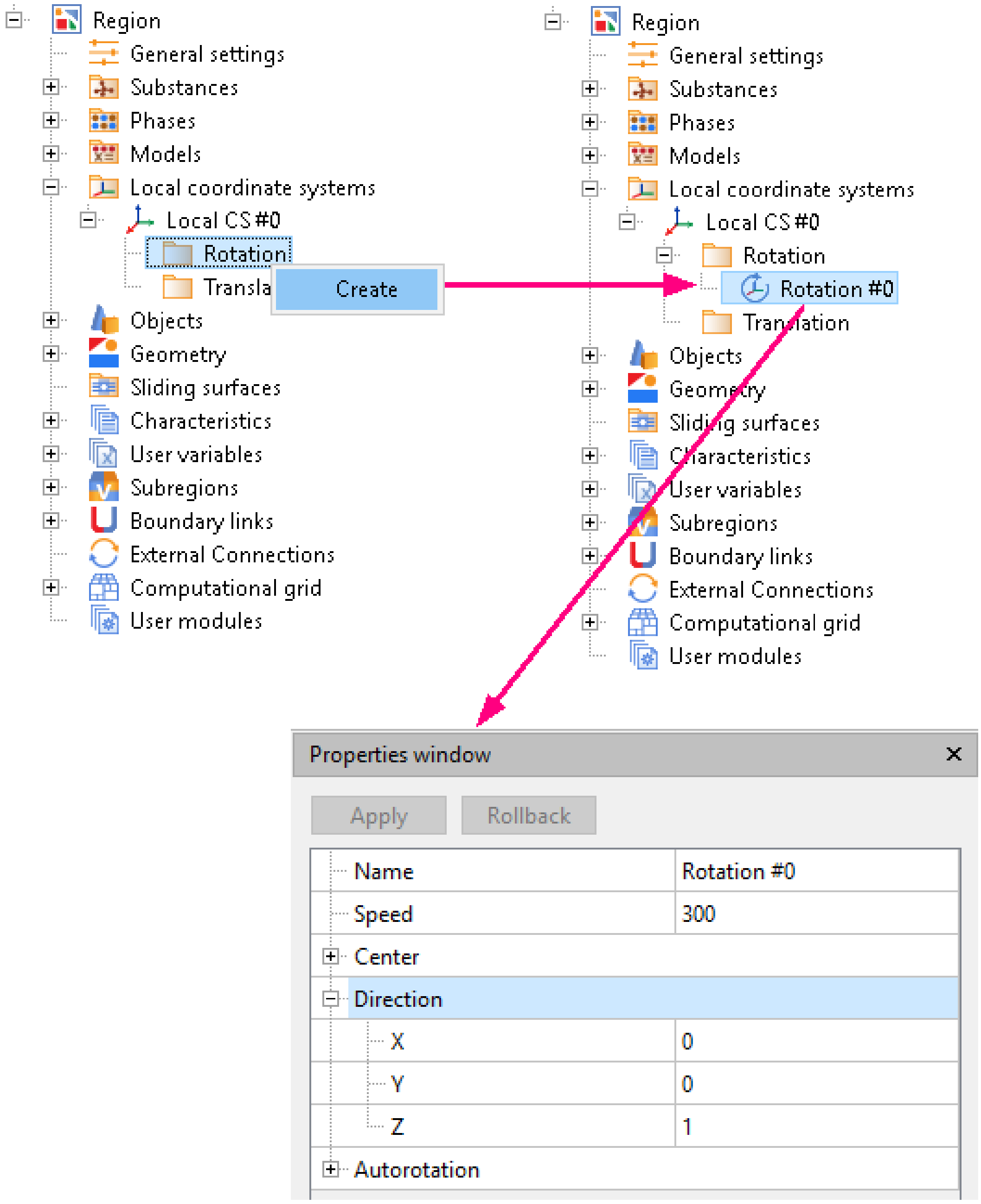
- From the context menu of the folder **Local coordinate systems**, select **Create**.



Specify a **Rotation**:

- In **Local CS #0** in the context menu of the folder **Rotation**, select **Create**.
- In the **Properties** window of **Rotation #0**, specify:

Speed	**= 300**	[radian s^{-1}]
Direction		
X	**= 0**	
Y	**= 0**	
Z	**= 1**	



To enable the accounting of the normal velocity's component during the rotation, specify in properties of **Region**:

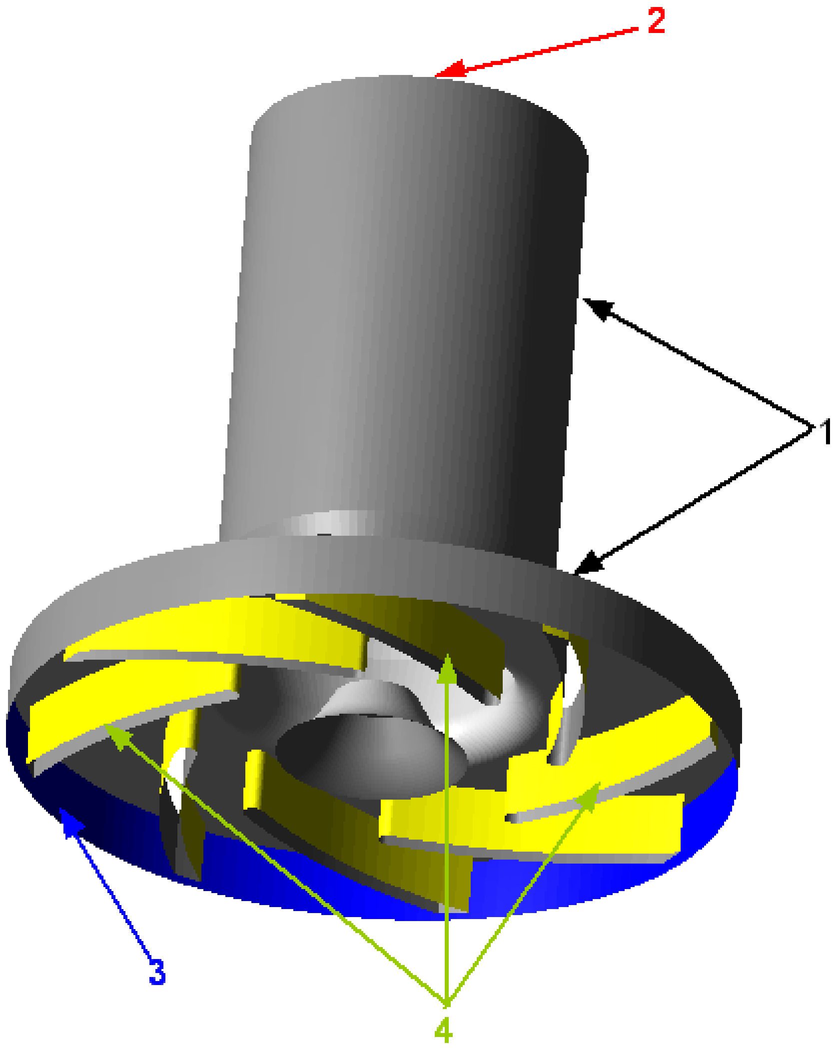
Local CS = **Local CS #0**

Rotation = **Rotation #0**

5.2.1.3 Boundary conditions

In the **Properties** window of the **SubRegion #0**, specify:

Model = Model #0



Specify the following boundary conditions:

Boundary 1

Name	**= Wall**	
Type	**= Wall**	
Variables		
Velocity (Phase #0)	**= Logarithm law**	
TurbEnergy (Phase #0)	**= Value in cell near wall**	
TurbDissipation (Phase #0)	**= Value in cell near wall**	

Boundary 2

Name	**= Inlet**	
Type	**= Inlet/Outlet**	
Variables		
Velocity (Phase #0)	**= Normal mass velocity**	
Mass velocity	**= 20**	[kg m^{-2} s^{-1}]
TurbEnergy (Phase #0)	**= Pulsations**	

Value	**= 0.01**	
TurbDissipation (Phase #0)	**= Turbulent scale**	
Value	**= 0.01**	[m]

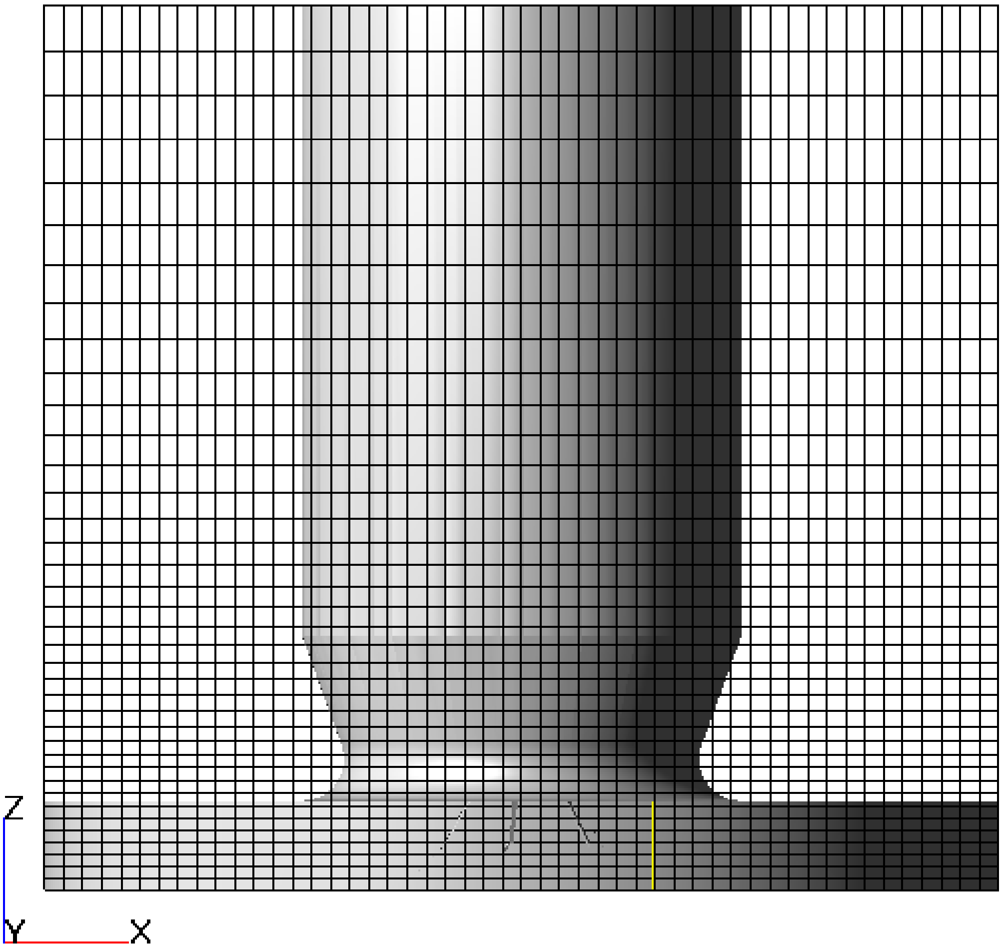
Boundary 3

Name	**= Outlet**	
Type	**= Free Outlet**	
Variables		
Velocity (Phase #0)	**= Pressure**	
Value	**= 0**	[Pa]
TurbEnergy (Phase #0)	**= Pulsations**	
Value	**= 0.01**	
TurbDissipation (Phase #0)	**= Turbulent scale**	
Value	**= 0.01**	[m]

Boundary 4

Name	**= Blades**
Type	**= Wall**
Local CS	**= Local CS #0**
Rotation	**= Rotation #0**
Variables	
Velocity (Phase #0)	**= Logarithm law**
TurbEnergy (Phase #0)	**= Value in cell near wall**
TurbDissipation (Phase #0)	**= Value in cell near wall**

5.2.1.4 Initial grid



In the **Properties** window of the **Initial grid**, click the button to open the **Initial grid editor**.

Specify in the **Initial grid editor**:

for axis OZ (click the button Z)

Grid parameters

h_max	**= 0.01**	[m]
h_min	**= 0.0025**	[m]

Specify **Reference line parameters** for the reference line with coordinate **z=0**:

h	**= 0.0025**	[m]
kh+	**= 1**	

Specify **Reference line parameters** for the reference line with coordinate **z=0.189**:

h	**= 0.01**	[m]
kh-	**= 1**	

Click **OK** to close the **Initial grid editor** with saving the entered data.

Specify in the **Properties** window of the **Initial grid**:

nX **= 50**

nY **= 50**

In the **Properties** window of the **Initial grid** click **Apply**.

5.2.1.5 Adaptation of the computational grid

Specify the adaptation of the computational grid on the boundary condition **Blades**:

- From the context menu of the folder **Computational grid > Adaptation**, select **Create**, so **Adaptation #0** will be created.
- From the context menu of the element **Adaptation #0 > Objects** select the command **Add/Remove Boundary Conditions** and, in the **Select boundary conditions** dialog box, place **SubRegion #0 : Blades** into the pane **Selected** and click **OK**.
- In the **Properties** window of the element **Adaptation #0** specify:

Enabled	**= Yes**
Max level N	**= 1**
Layers	
Layers for Level N	**= 4**

5.2.1.6 Parameters of calculation

Specify in the **Solver** tab in properties of the **Time step** element:

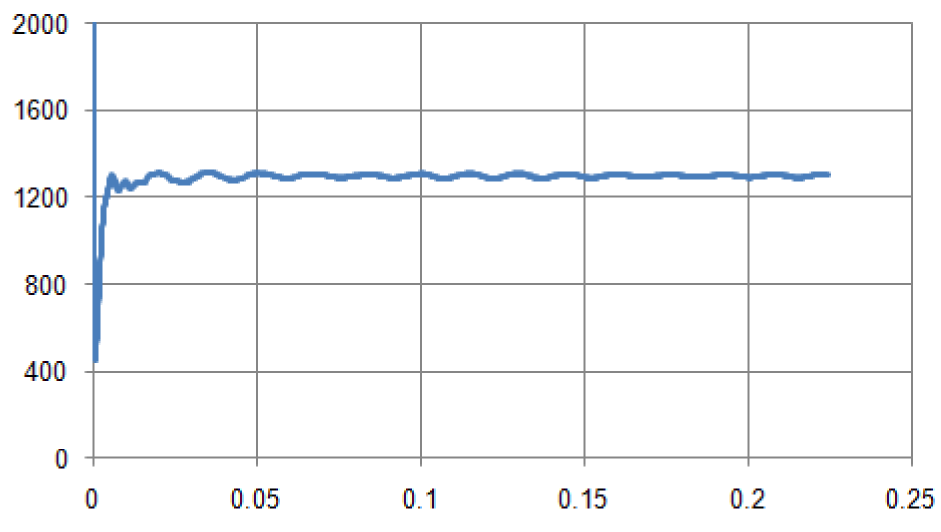
Method	**= In seconds**	
Constant step	**= 0.0001**	[s]

5.2.1.7 Visualization

To view the dynamics of the solution during the computation, prior to computation, specify visualization of:

1. Pressure variation on inlet
2. Velocity distribution in the plane of rotation of the blades

5.2.1.7.1 Pressure variation on inlet



- Create a **Supergroup** on the BC **Inlet** using the command **Create supergroup > In Preprocessor** from the context menu.
- Create **Characteristics** on this **Supergroup**.
- In the **Properties** window of the appropriate **Characteristics** object, which locates in the **Preprocessor** tab of the project tree, specify:

Variable	
Variable	**= Pressure**

- In the **Properties** window of the appropriate **Characteristics** object, which locates in the **Postprocessor** tab of the project tree, specify:

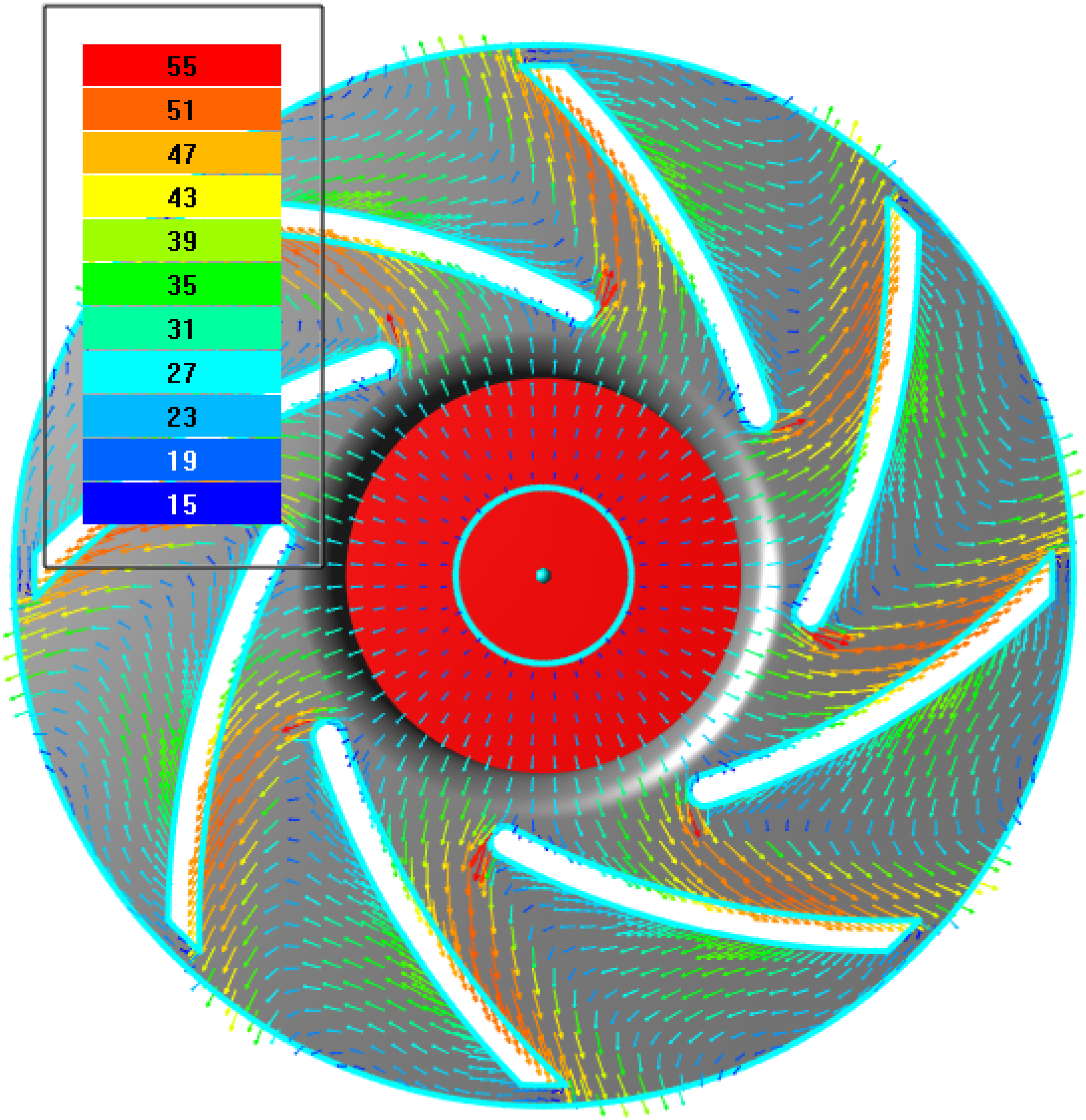
  Save to file

  Type = **Automatic**

- After the computation is finished, open the `glo`-file, which is recorded by the data from the **Characteristics**, and plot the dependency of **Avg** by **Time**.

5.2.1.7.2 Velocity distribution

Visualization at the step number 2250:



- In properties of **Plane #0** specify:

Object

Reference point

X = **0**

Y = **0**

Z = **0.01**

Normal

X = **0**

Y = **0**

Z = **1**

(to direct a **Plane**'s normal along the axis **Z**, you can also click the **Operations >**  button in the **Properties** window of the **Plane**)

- Create a layer **Vectors** on **Plane #0**.
- In properties of the **Vectors** specify:

On regular grid = **No**

Coloring

  Variable

    Variable = **Velocity**

  Value range

    Mode = **Manual**

    Max = **55**

    Min = **15**

  Palette

    Appearance

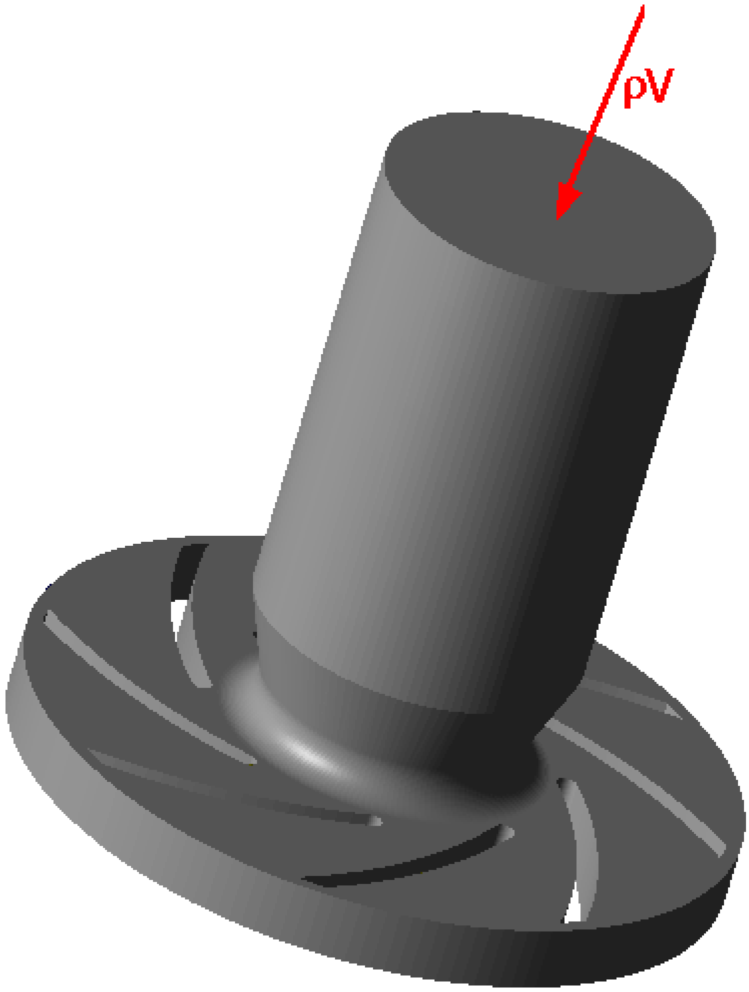
      Enabled = **Yes**

      Style = **Style 1**

The program will automatically specify the variable, which is used to build the vectors, **Variable > Variable = Velocity**.

5.2.2 Sector of a rotor

This example illustrates simulation of movement of incompressible gas between blades of a 8-blade rotor. To speed up the calculation, the flow is simulated in only one sector of the rotor.



Parameters of the problem setting

Dimensions:

Length of the inlet passage	L	= 0.17	[m]
Radius of the rotor	R	= 0.1	[m]

Inflow parameters:

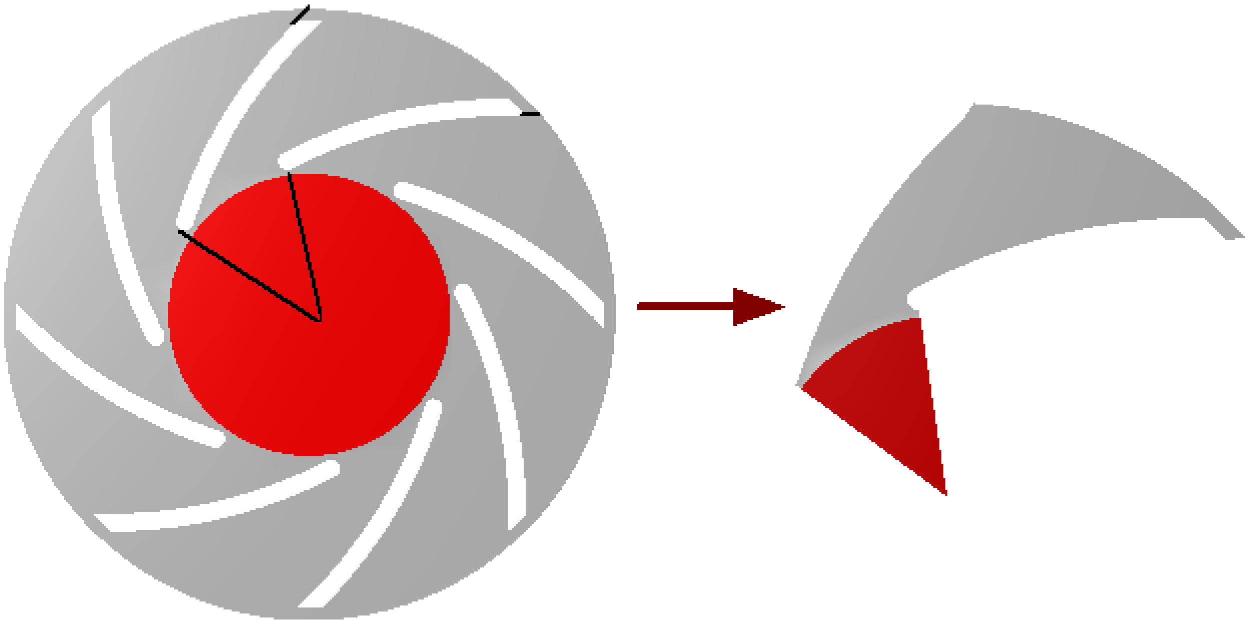
Mass velocity on inlet	ρV_{inl}	= 20	[kg m^{-2} s^{-1}]
Speed of rotation	ω	= 50	[radian s^{-1}]

Substance: = Air

Geometry: **`RotorSector.wrl`**

Project: **RotorSector**

5.2.2.1 Making geometry of the computational domain



When simulating the flow in the rotor consisting of several identical sectors, to reduce the dimension of the task, it is reasonable to calculate the motion in one of the sectors. Do the following:

- At the step of preparation the geometry, cut from the full geometric model of the rotor one of its periodic sectors.
- When specifying the project, match appropriate surfaces using a periodic binder condition.

5.2.2.2 Physical model

In the folder **Substances**:

- Create **Substance #0**.
- Load **Substance #0** from the **Standard** substance database:
    - From the context menu of **Substance #0** select **Load from SD > Standard**.
    - In the new window **Load from database**, select:

Substances	**= Air**
Phases	**= Gas (equilibrium)**

In the folder **Phases**:

- Create continuous **Phase #0**.
- Add the **Air_Gas (equilibrium)** substance into the folder **Phase #0 > Substances**.
- In properties of the folder **Phase #0 > Physical processes**, specify:

Motion	**= Navier-Stokes model**
Turbulence	**= KES**

In the folder **Models**:

- Create **Model #0**.
- Add **Phase #0** into subfolder **Model #0 > Phases**.
- In the folder **Init. data #0**, specify:

Pulsations(Phase #0)

Value	**= 0.01**	
Turbulent scale(Phase #0)		
Value	**= 0.01**	[m]

5.2.2.3 Rotation

Create a **Local coordinate system** (local coordinate system):

- From the context menu of the folder **Local coordinate systems**, select **Create**.

Specify a **Rotation**:

- In the folder **Local CS #0 > Rotation** create **Rotation #0.**
- In the **Properties** window of **Rotation #0**, specify:

Speed	**= 50**	[radian s^{-1}]
Direction		
X	**= 0**	
Y	**= 0**	
Z	**= 1**	

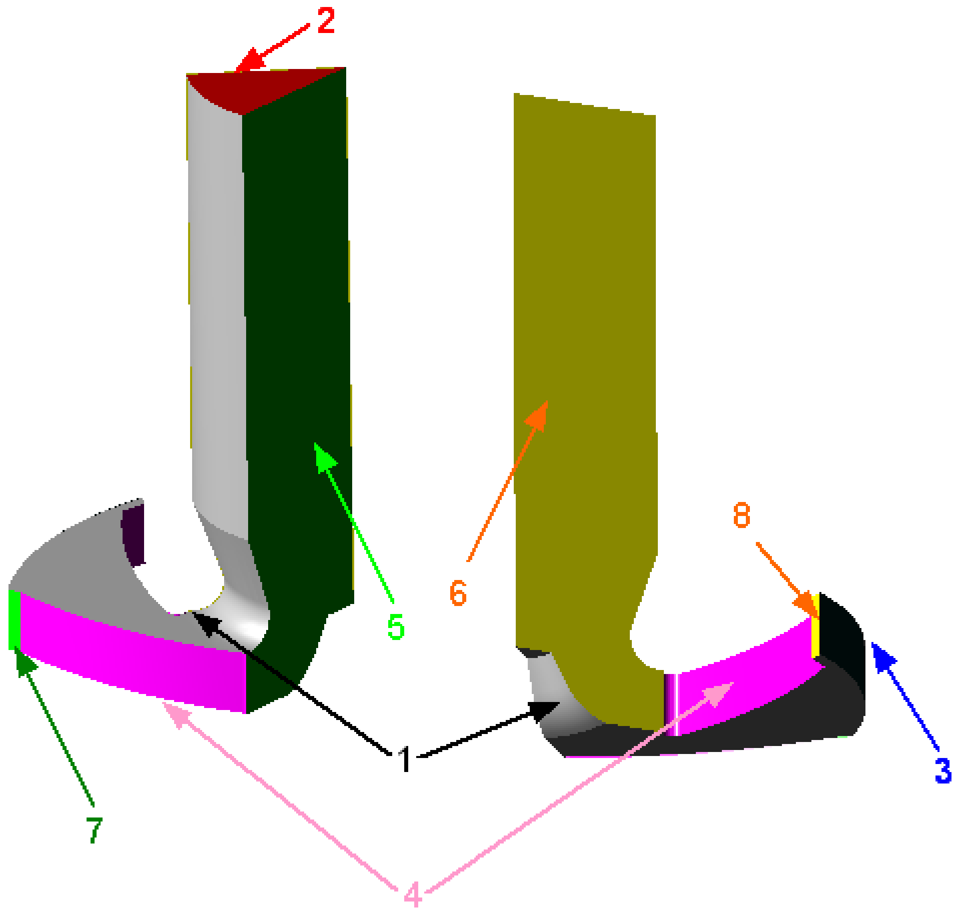
In the **Properties** window of **Region**, specify:

Local CS	**= Local CS #0**
Rotation	**= Rotation #0**

5.2.2.4 Boundary conditions

In properties of **SubRegion #0** specify:

Model = Model #0



Specify the following boundary conditions:

Boundary 1

Name	**= Wall**	
Type	**= Wall**	
Variables		
Velocity (Phase #0)	**= Logarithm law**	
TurbEnergy (Phase #0)	**= Value in cell near wall**	
TurbDissipation (Phase #0)	**= Value in cell near wall**	

Boundary 2

Name	**= Inlet**	
Type	**= Inlet/Outlet**	
Variables		
Velocity (Phase #0)	**= Normal mass velocity**	
Mass velocity	**= 20**	[kg m^{-2} s^{-1}]
TurbEnergy (Phase #0)	**= Pulsations**	
Value	**= 0.01**	
TurbDissipation (Phase #0)	**= Turbulent scale**	
Value	**= 0.01**	[m]

Boundary 3		
Name	**= Outlet**	
Type	**= Free Outlet**	
Variables		
Velocity (Phase #0)	**= Pressure**	
Value	**= 0**	[Pa]
TurbEnergy (Phase #0)	**= Pulsations**	
Value	**= 0.01**	
TurbDissipation (Phase #0)	**= Turbulent scale**	
Value	**= 0.01**	[m]

Boundary 4	
Name	**= Blades**
Type	**= Wall**
Local CS	**= Local CS #0**
Rotation	**= Rotation #0**
Variables	
Velocity (Phase #0)	**= Logarithm law**
TurbEnergy (Phase #0)	**= Value in cell near wall**
TurbDissipation (Phase #0)	**= Value in cell near wall**

Boundary 5	
Name	**= Binder surface 1_1**
Type	**= Connected**

Boundary 6	
Name	**= Binder surface 1_2**
Type	**= Connected**

Boundary 7	
Name	**= Binder surface 2_1**
Type	**= Connected**

Boundary 8	
Name	**= Binder surface 2_2**
Type	**= Connected**

5.2.2.5 Binding the subregions

When binding boundary conditions that are not different sides of one surface, it is necessary to specify **Snap points**, which are reference points on each of the two bound surfaces matching to each other. It is enough to specify one pair of snap points for each binder.

To specify a pair of snap points, it is necessary:

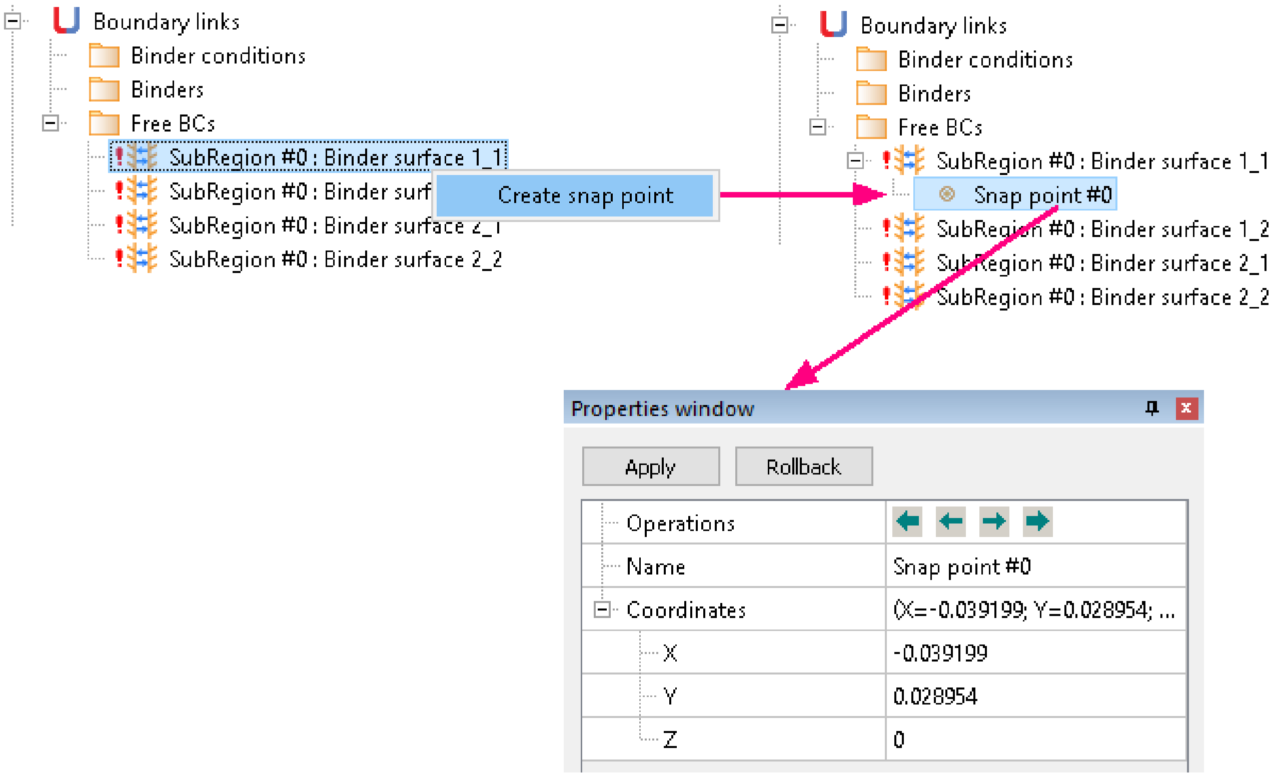
- In the folder **Free BCs** in the context menus of both connected boundary conditions select **Create snap point**.
- In the **Properties** windows of both snap points specify their coordinates in such a way that provides correct matching of the surfaces that will be bound.

Create a snap point on the boundary condition **Binder surface 1_1**:

- From the context menu of the element **Free BCs > SubRegion #0 : Binder surface 1_1** select the command **Create snap point**.
- In the **Properties** window of **Snap point #0**, specify:

Coordinates

X	**= -0.039199**
Y	**= 0.028954**
Z	**= 0**



Create a snap point on the BC **Binder surface 1_2**:

- From the context menu of the BC **Binder surface 1_2** select **Create snap point**.
- In the **Properties** window of **Snap point #0**, specify:

Coordinates

X	**= -0.007244**
Y	**= 0.048191**
Z	**= 0**

Create a snap point on the BC **Binder surface 2_1**:

- From the context menu of the BC **Binder surface 2_1** select **Create snap point**.
- In the **Properties** window of **Snap point #0**, specify:

Coordinates

X	**= 0.004345**
Y	**= 0.097904**
Z	**= 0**

Create a snap point on the BC **Binder surface 2_2**:

- From the context menu of the BC **Binder surface 2_2** select **Create snap point**.
- In the **Properties** window of **Snap point #0**, specify:

Coordinates

X	**= 0.072301**
Y	**= 0.066156**
Z	**= 0**

Create **Binder #0** and **Binder #1**:

- From the context menu of the folder **Binders**, select **Create all**.

Create **Binder condition #0**:

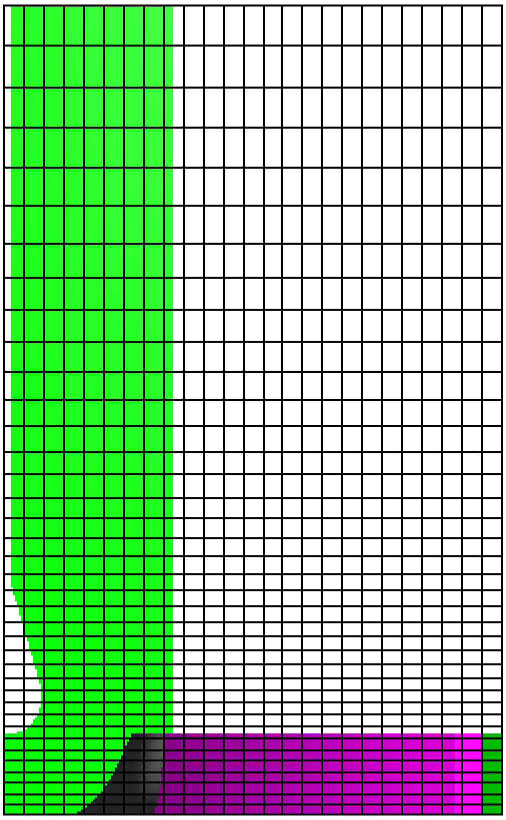
- From the context menu of the folder **Binder conditions**, select **Create**.
- In the **Create binder condition** dialog box, which opens, specify:

Connection type	**= Periodic surface**
1st model	**= Model #0**
2nd model	**= Model #0**

Specify the matching between the **Binders** and the **Binder condition**:

- From the context menu of the folder **Binder conditions > Binder condition #0 > Binders**, select **Add/Remove** and click **Add All** button in the **Select binders** dialog box, which opens.

5.2.2.6 Initial grid



In the **Properties** window of the **Initial grid**, click the button to open the **Initial grid editor**.

Specify in the **Initial grid editor**:

for axis OZ (click the button )

Grid parameters

h_max	**= 0.01**	[m]
h_min	**= 0.0025**	[m]

Specify **Reference line parameters** for the reference line with coordinate **z=0**:

h	**= 0.0025**	[m]
kh+	**= 1**	

Specify **Reference line parameters** for the reference line with coordinate **z=0.189**:

h	**= 0.01**	[m]
kh-	**= 1**	

Click **OK** to close the **Initial grid editor** with saving the entered data.

Specify in the **Properties** window of the **Initial grid**:

nX	**= 25**
nY	**= 25**

In the **Properties** window of the **Initial grid** click **Apply**.

5.2.2.7 Adaptation of the computational grid

Specify the adaptation of the computational grid on the boundary condition **Blades**:

- From the context menu of the folder **Computational grid > Adaptation**, select **Create**, so **Adaptation #0** will be created.
- From the context menu of the element **Adaptation #0 > Objects** select the command **Add/Remove Boundary Conditions** and, in the **Select boundary conditions** dialog box, place **SubRegion #0 : Blades** into **Selected** and click **OK**.
- In the **Properties** window of the element **Adaptation #0** specify:

Enabled	**= Yes**
Max level N	**= 1**
Layers	
Layers for Level N	**= 2**

5.2.2.8 Parameters of calculation

Specify in the **Solver** tab in properties of the **Time step** element:

Method	**= In seconds**	
Constant step	**= 0.0001**	[s]

5.2.2.9 Visualization

To view the dynamics of the solution during the computation, prior to computation, specify visualization of:

1. **Pressure variation on inlet**
2. **Velocity distribution** in the plane of rotation of the blades

5.2.2.9.1 Pressure variation on inlet

We consider below two methods of displaying a plot the pressure variation on inlet. The first steps of these methods are same:

- On the BC **Inlet** create a **Supergroup** in **Preprocessor** (use in the context menu the command **Create supergroup > In Preprocessor**).
- Create **Characteristics** on this **Supergroup**.
- In properties of the created element **Characteristics** > **Characteristics #0 (Supergroup on "Inlet")** specify:

Variable

Variable **= Pressure**

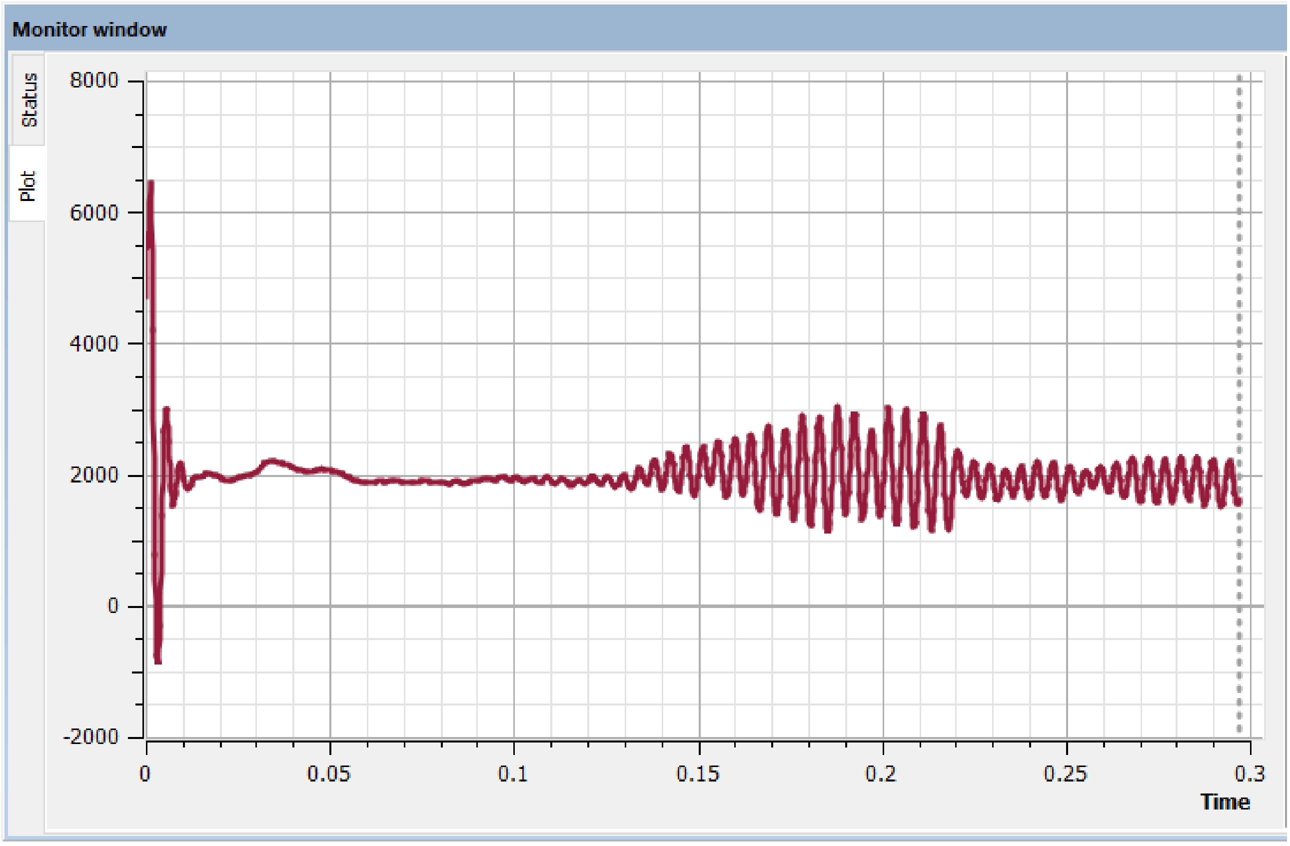
Displaying the plot in the Monitor window

In the **Solver** tab, in the folder **Stopping conditions > User values** create a user **Stop criterion #0** and in its properties specify:

Object **= Characteristics #0 (Supergroup on "Inlet")**

Variable **= <f surf.>**

Run the computation and view dynamics of the pressure variation on inlet in the **Plot** tab of the **Monitor** window (select the **Stop criterion #0** in the right pane and set parameters of the plot if necessary):



RotorSector_PressureVariatinChart_MonYellow__E.png

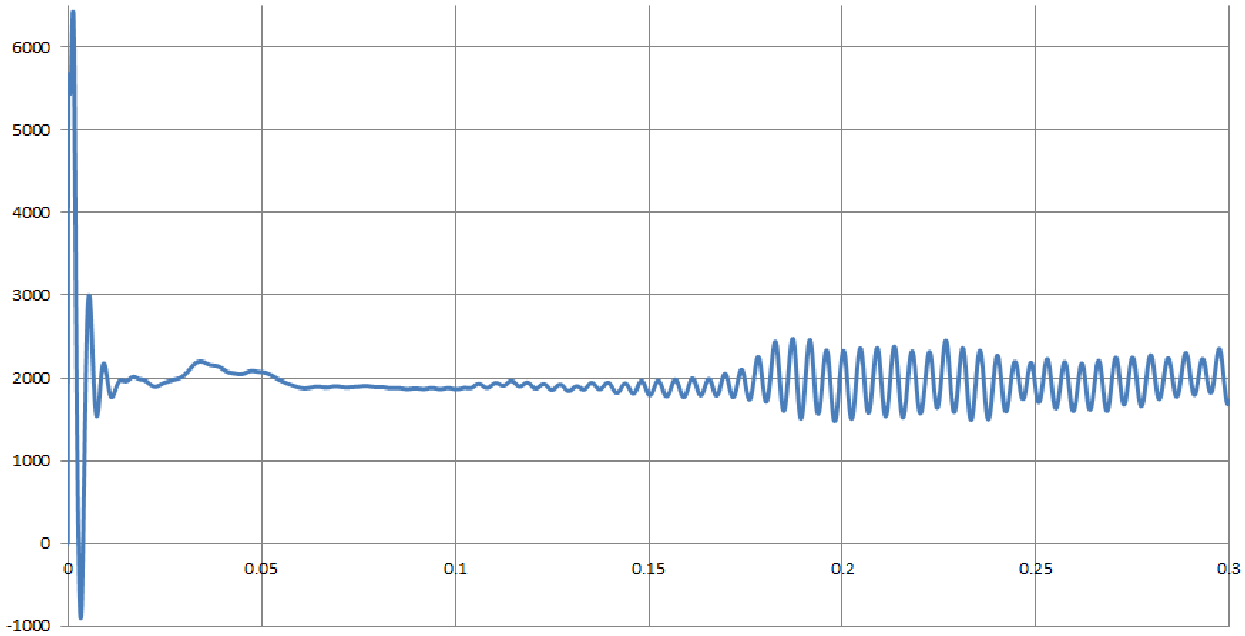
Displaying the plot in an external program

- In the **Postprocessor** tab, in properties of the element **Characteristics** > **Characteristics #0 (Supergroup on "Inlet")**, specify:

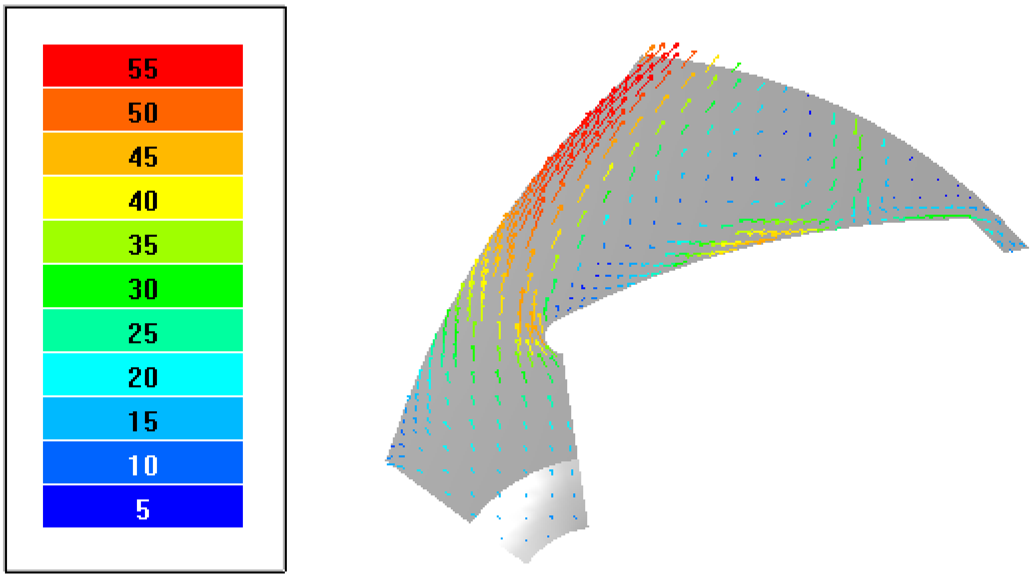
Save to file

Type **= Automatic**

- After the computation is finished, open the `glo`-file, which is specified in properties of the element **Characteristics #0 (Supergroup on "Inlet")**, by an external program (for example, by *Excel*) and plot the dependency of **Avg** by **Time**.



5.2.2.9.2 Velocity distribution



- In the **Properties** window of **Plane #0** specify:

Object

Reference point		
	X	**= 0**
	Y	**= 0**
	Z	**= 0.01**
Normal		
	X	**= 0**
	Y	**= 0**
	Z	**= 1**

(to direct a **Plane**'s normal along the axis **Z**, you can also click the **Operations >** Z↓ button in the **Properties** window of the **Plane**)

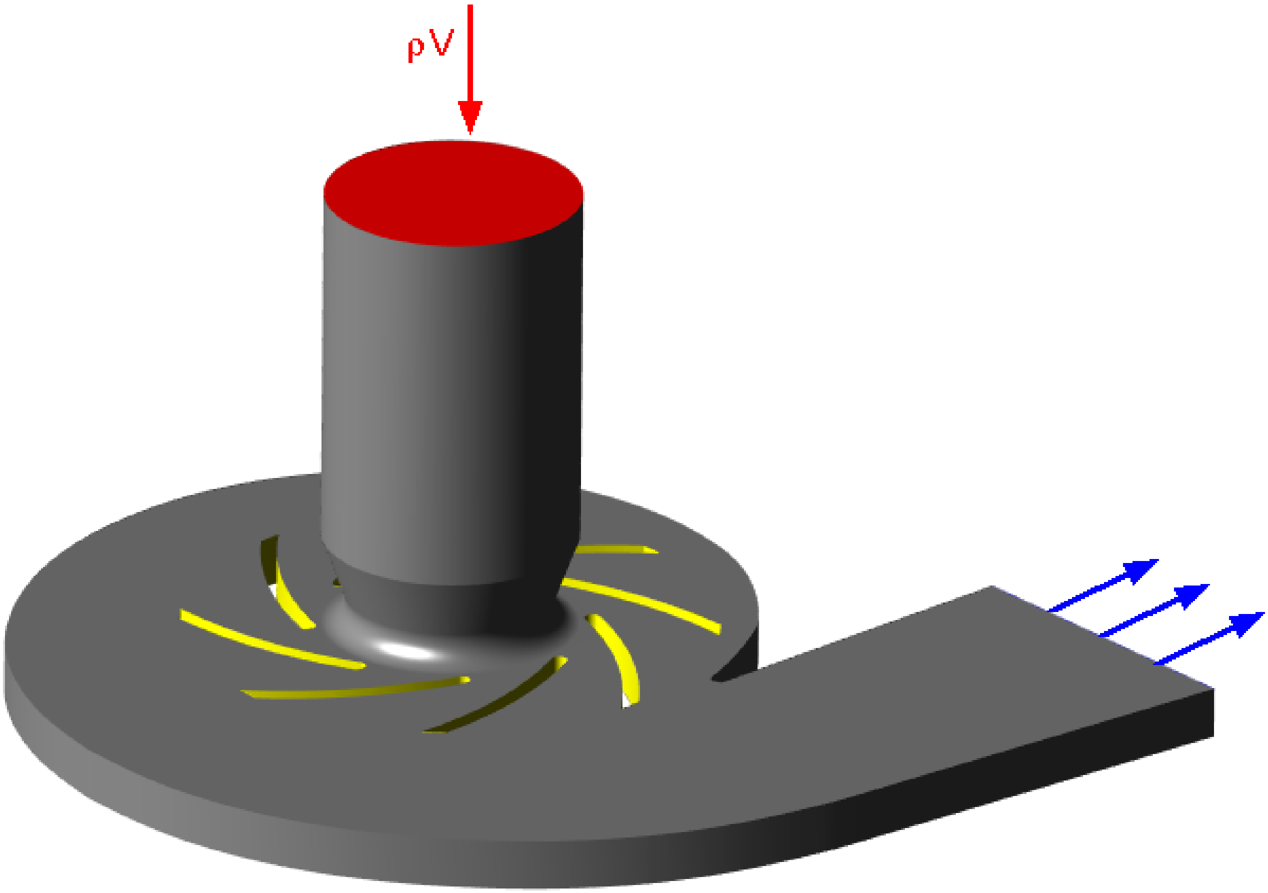
- Create a layer **Vectors** on **Plane #0**.
- In the **Properties** window of the layer specify:

On regular grid	**= No**
Coloring	
Variable	
Variable	**= Velocity**
Value range	
Mode	**= Manual**
Max	**= 55**
Min	**= 5**
Palette	
Appearance	
Enabled	**= Yes**
Style	**= Style 1**

The program will automatically specify the variable, which is used to build the vectors, **Variable > Variable = Velocity**.

5.2.3 Rotor+Stator

This example illustrates simulation of movement of the air between blades of a rotating rotor and in the stator's outlet passage.



Parameters of the problem setting

Dimensions:

Length of the inlet passage	L	= 0.17	[m]
Radius of the rotor	R	= 0.1	[m]

Inflow parameters:

Mass velocity on inlet	ρV_{inl}	= 20	[kg m^{-2} s^{-1}]
Speed of rotation	ω	= 50	[radian s^{-1}]

Substance: = Air

Geometry: `RotorStator.STL`

Project: `RotorStator`

5.2.3.1 Making geometry of the computational domain

When simulating rotations of surfaces in the computational domain, the outer boundary of which is not a surface of revolution around an appropriate axis of rotation, it is necessary to split the geometry into several subregions. Thus, the outer boundaries of the subregions, where the rotation is defined, must be surfaces of revolution around an appropriate axis. Relationship between subregions is provided by **Sliding Binder conditions** that allow simulation of fluid flows across subregions' borders taking into account the rotation of the rotating subregion relating to the stationary subregion.

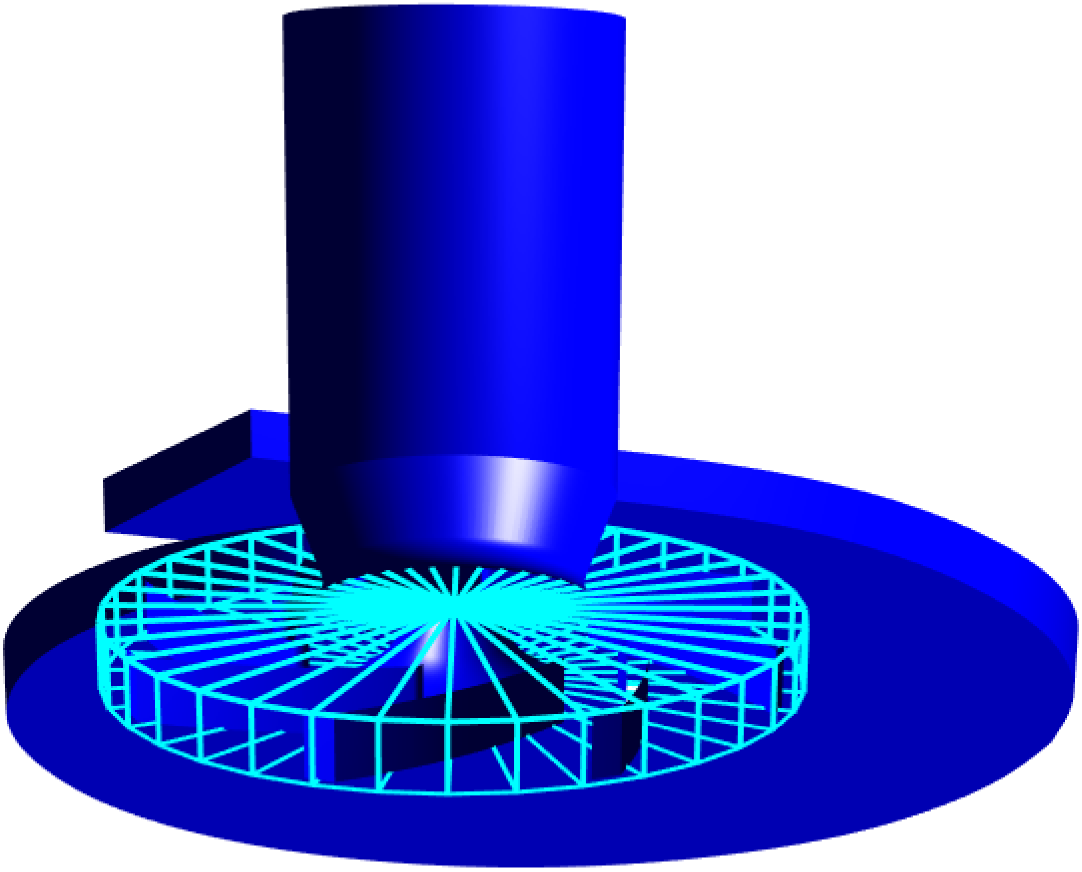
To split subregions, you have to do the following steps:

- Create a **Rotation**.
- Create a splitting surface.
- Split the computational domain into **Subregions** by the splitting surface.

Create a **Rotation**:

- Create a **Local coordinate system**.
- In the folder **Local CS #0 > Rotation** create **Rotation #0**.
- In the **Properties** window of **Rotation #0**, specify:

Speed	**= 50**	[radian s^{-1}]
Direction		
X	**= 0**	
Y	**= 0**	
Z	**= 1**	

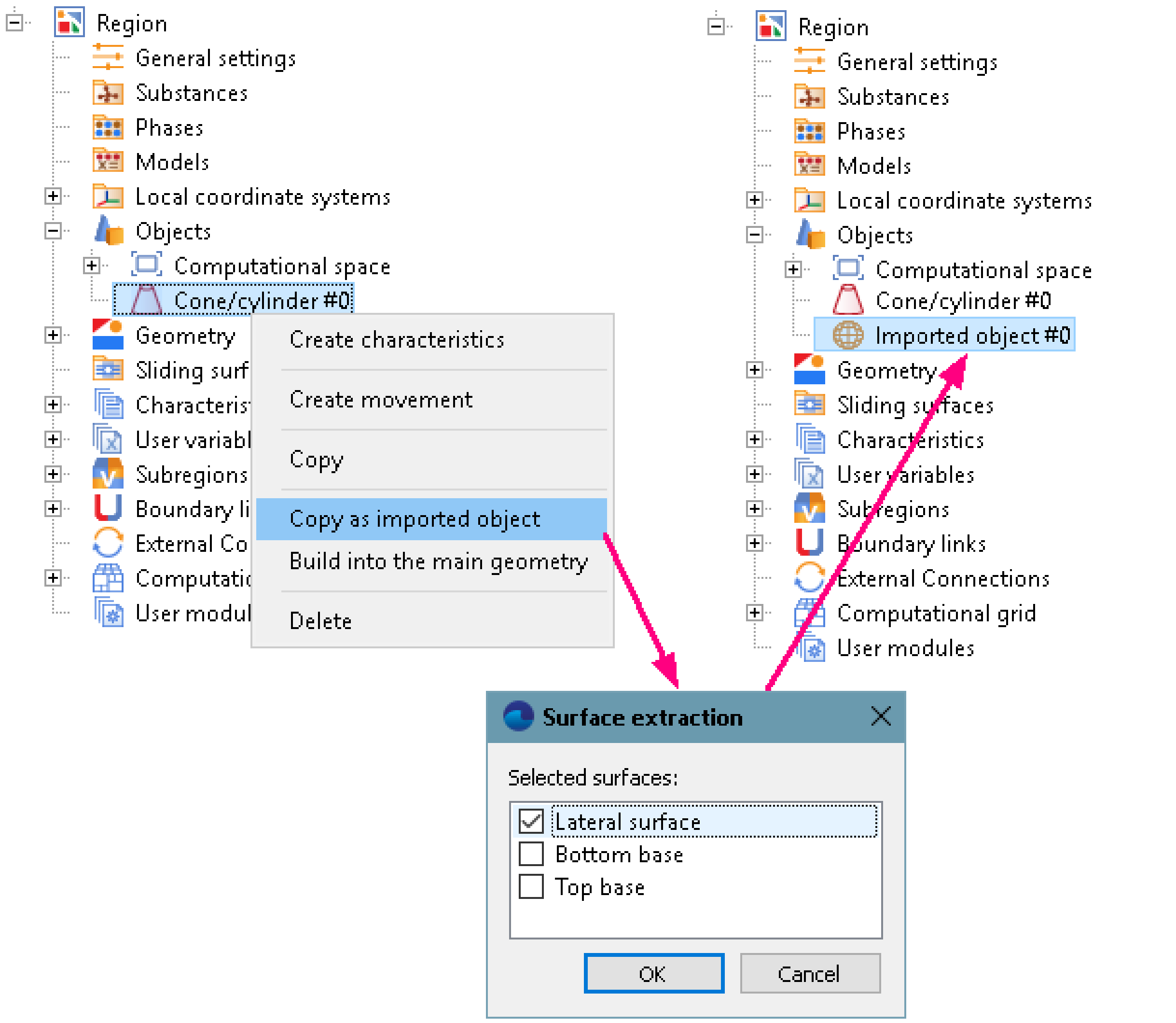


Create a separating surface:

- In the folder **Object** create a **Cone/cylinder**.
- Specify in the **Properties** window of **Cone/cylinder #0**.

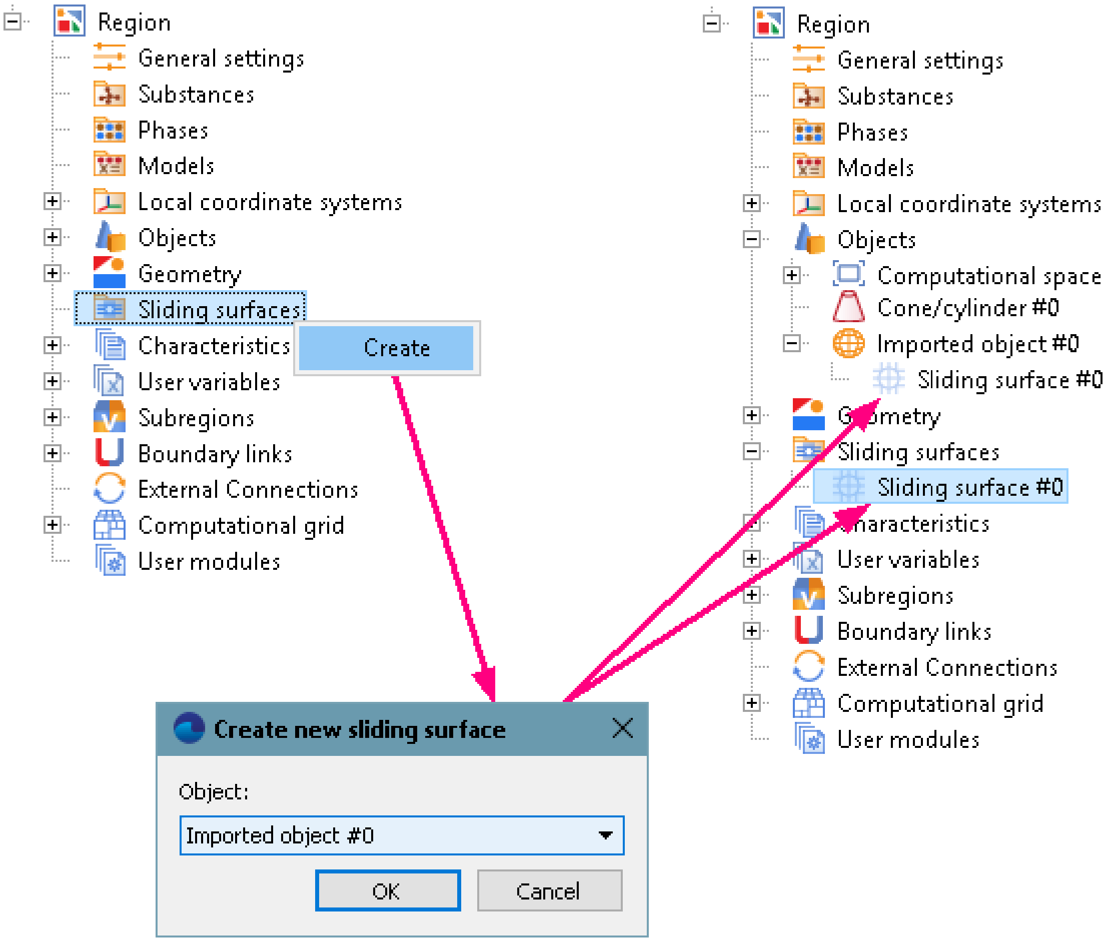
Location		
Reference point		
X	**= 0.0005**	[m]
Y	**= 0**	[m]
Z	**= 0**	[m]
Axis X		
X	**= 0**	
Y	**= 0**	
Z	**= 1**	
Parameters		
Height	**= 0.018**	[m]
Radius 1	**= 0.1025**	[m]
Radius 2	**= 0.1025**	[m]
Base ratio	**= 1**	

- From the context menu of **Cone/cylinder #0** select **Copy as imported object**.
- In the **Surface extraction** window, check **Lateral surface** only. **Bottom base** and **Top base** must remain unchecked.



* From the context menu of the **Sliding surfaces** folder, select **Create**.
* In the **Create new sliding surface** window, specify:

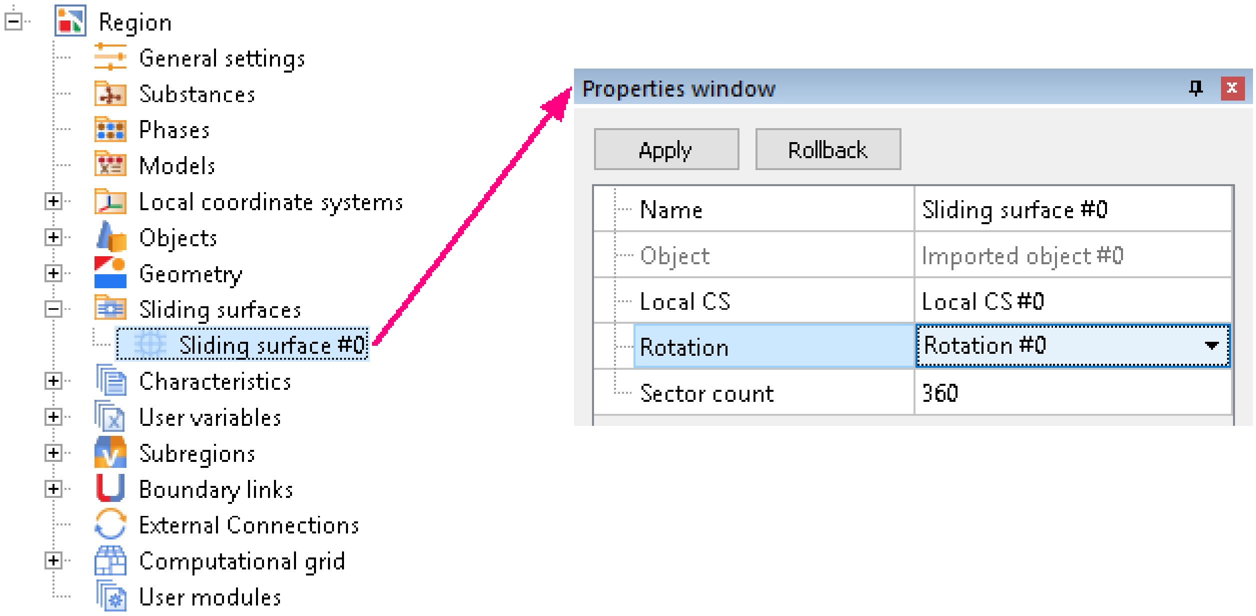
Object = **Imported object #0**



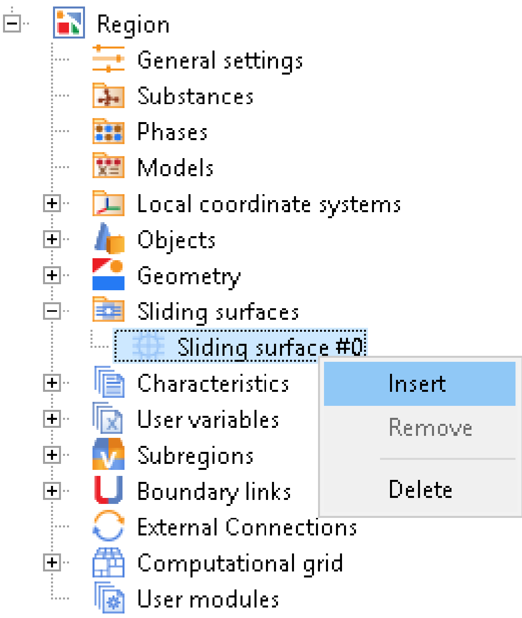
Split the computational domain into **Subregions** by the splitting surface:

- In the **Properties** window of **Sliding surface #0** specify:

Local CS	**= Local CS #0**
Rotation	**= Rotation #0**



- From the context menu of **Sliding surface #0**, select **Insert**:



5.2.3.2 Physical model

In the folder **Substances**:

- Create **Substance #0**.
- Load **Substance #0** from the **Standard** substance database:
  - From the context menu of **Substance #0** select **Load from SD > Standard.**
  - In the new window **Load from database**, select:

Substances	**= Air**
Phases	**= Gas (equilibrium)**

In the folder **Phases**:

- Create a continuous **Phase #0**.
- Add the **Air_Gas (equilibrium)** substance into the folder **Phase #0 > Substances**.
- Specify in properties of the folder **Phase #0 > Physical processes**:

Motion	**= Navier-Stokes model**
Turbulence	**= KES**

In the folder **Models**:

- Create **Model #0**.
- Add **Phase #0** into subfolder **Model #0 > Phases**.
- In the folder **Init. data #0**, specify:

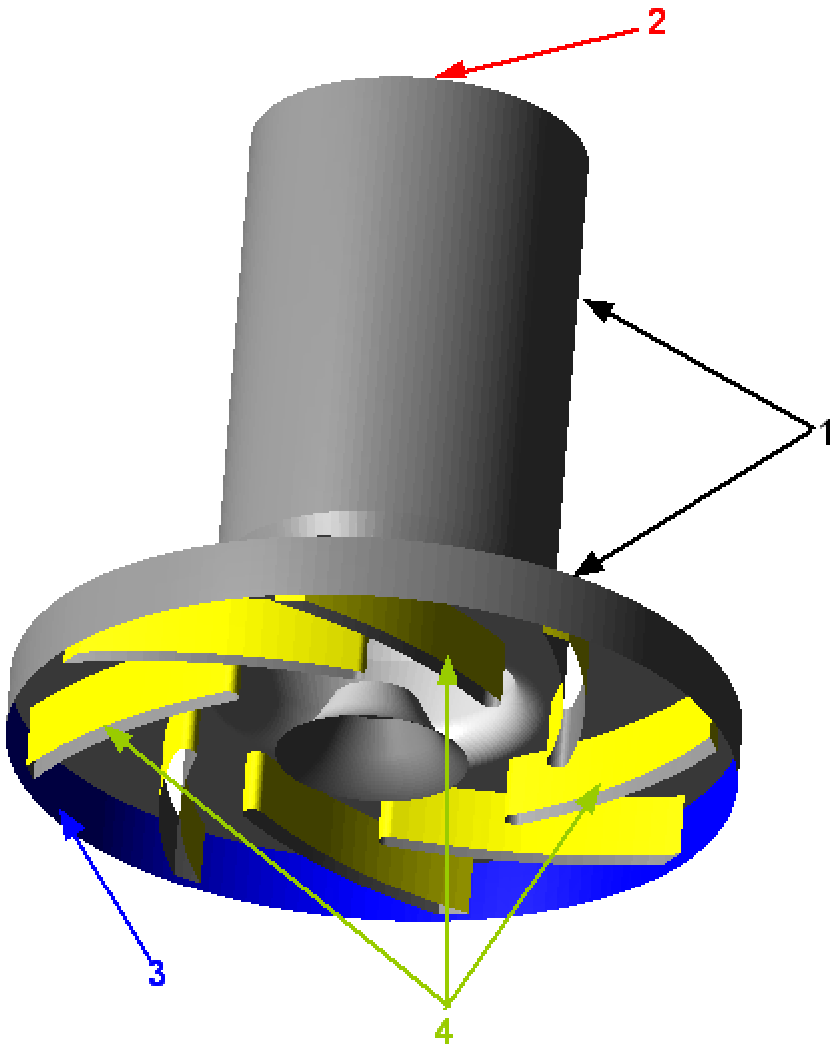
Pulsations(Phase #0)			
	Value	**= 0.01**	
Turbulent scale(Phase #0)			
	Value	**= 0.01**	[m]

5.2.3.3 Boundary conditions

In the **Properties** window of the **Subregion**, which corresponds to the rotor (see illustration below), specify:

Name **= Rotor**

Model = **Model #0**



Specify the following boundary conditions:

Boundary 1

Name	**= Wall**
Type	**= Wall**
Variables	
Velocity(Phase #0)	**= Logarithm law**
TurbEnergy(Phase #0)	**= Value in cell near wall**
TurbDissipation(Phase #0)	**= Value in cell near wall**

Boundary 2

Name	**= Inlet**	
Type	**= Inlet/Outlet**	
Variables		
Velocity(Phase #0)	**= Normal mass velocity**	
Mass velocity	**= 20**	[kg m^{-2} s^{-1}]
TurbEnergy(Phase #0)	**= Pulsations**	
Value	**= 0.01**	

TurbDissipation(Phase #0)	**= Turbulent scale**	
Value	**= 0.01**	[m]

Boundary 3

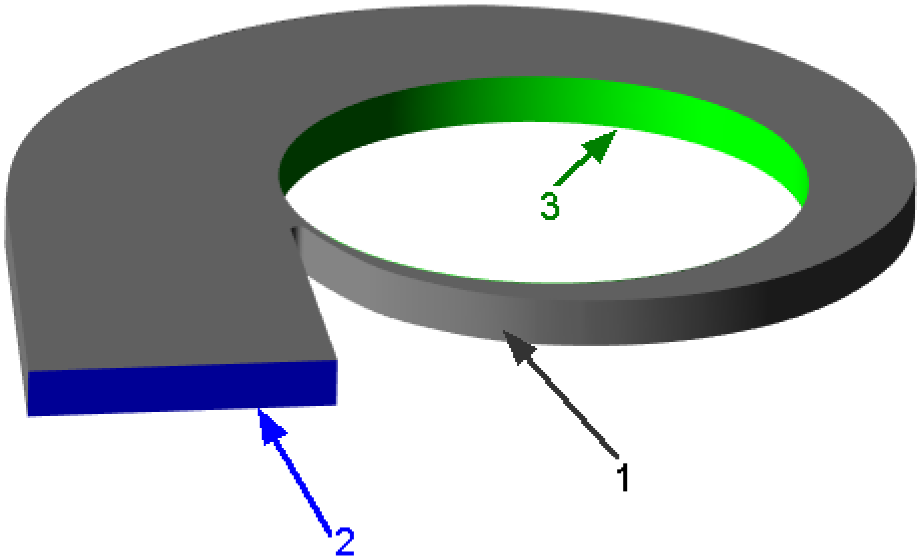
Name	**= Connection surface**
Type	**= Connected**

Boundary 4

Name	**= Blades**
Type	**= Wall**
Local CS	**= Local CS #0**
Rotation	**= Rotation #0**
Variables	
Velocity(Phase #0)	**= Logarithm law**
TurbEnergy(Phase #0)	**= Value in cell near wall**
TurbDissipation(Phase #0)	**= Value in cell near wall**

In the **Properties** window of the **Subregion**, which corresponds to the stator (see illustration below), specify:

Name	**= Stator**
Model	**= Model #0**



Specify the following boundary conditions:

Boundary 1

Name	**= Wall**
Type	**= Wall**
Variables	
Velocity(Phase #0)	**= Logarithm law**
TurbEnergy(Phase #0)	**= Value in cell near wall**
TurbDissipation(Phase #0)	**= Value in cell near wall**

Boundary 2

Name	**= Outlet**	
Type	**= Free Outlet**	
Variables		
Velocity(Phase #0)	**= Pressure**	
Value	**= 0**	[Pa]
TurbEnergy(Phase #0)	**= Pulsations**	
Value	**= 0.01**	
TurbDissipation(Phase #0)	**= Turbulent scale**	
Value	**= 0.01**	[m]

Boundary 3

Name	**= Connection surface**
Type	**= Connected**

5.2.3.4 Binding the subregions

Create **Binder #0**:

- Select **Create all** from the context menu of the folder **Binders**.

Create **Binder condition #0**:

- Select **Create** from the context menu of the folder **Binder conditions**.
- In the **Create binder condition** dialog box, which opens, specify:

Connection type	**= Sliding surface**
1st model	**= Model #0**
2nd model	**= Model #0**

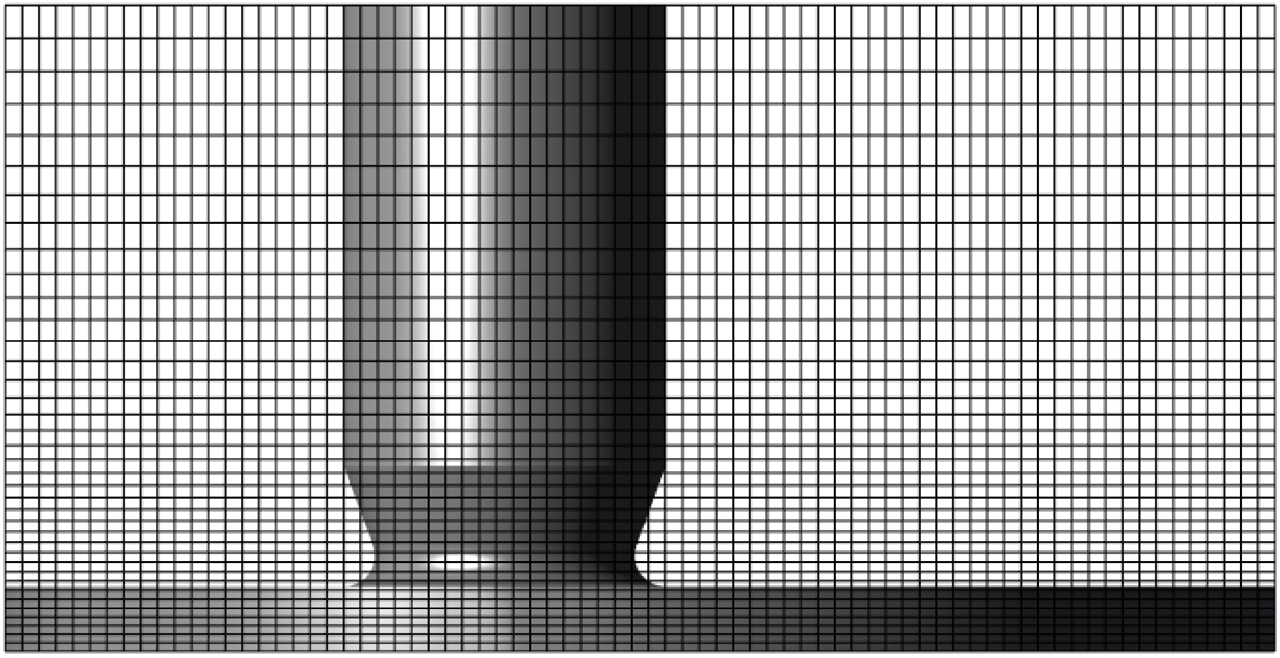
Specify the matching between the **Binder** and the **Binder condition**:

- From the context menu of the folder **Binder condition #0 > Binders** select **Add/Remove**.
- Add **Binder #0** into the folder.

In the **Properties** window of the subregion **Rotor**, specify:

Local CS	**= Local CS #0**
Rotation	**= Rotation #0**

5.2.3.5 Initial grid



In the **Properties** window of the **Initial grid**, click the button to open the **Initial grid editor**.

Specify in the **Initial grid editor**:

for axis OZ (click the button Z)

Grid parameters

h_max	**= 0.01**	[m]
h_min	**= 0.0025**	[m]

Specify **Reference line parameters** for the reference line with coordinate **z=0**:

h	**= 0.0025**	[m]
kh+	**= 1**	

Specify **Reference line parameters** for the reference line with coordinate **z=0.189**:

h	**= 0.01**	[m]
kh-	**= 1**	

Click **OK** to close the **Initial grid editor** with saving the entered data.

Specify in the **Properties** window of the **Initial grid**:

nX	**= 75**
nY	**= 70**

In the **Properties** window of the **Initial grid** click **Apply**.

5.2.3.6 Adaptation of the computational grid

Create an **Adaptation** of the computational grid that will be active in the subregion **Rotor** (on **Blades** and on the **Connection surface**) and in the subregion **Stator** on the **Connection surface**:

- From the context menu of the folder **Computational grid > Adaptation**, select **Create**, so **Adaptation #0** will be created.
- From the context menu of the element **Adaptation #0 > Subregions** select **Add/Remove** and, in the **Select Subregions** dialog box, place both **Rotor** and **Stator** into the **Selected** pane, and then click **OK**.
- From the context menu of the element **Adaptation #0 > Objects** select **Add/Remove Boundary Conditions** and, in the **Select boundary conditions** dialog box, place **Rotor : Blades**, **Rotor : Connection surface**, and **Stator : Connection surface** into the **Selected** pane, and then click **OK**.

- In the **Properties** window of the element **Adaptation #0** specify:

Enabled	**= Yes**
Max level N	**= 1**
Layers	
Layers for Level N	**= 2**

5.2.3.7 Parameters of calculation

Specify in the **Solver** tab in properties of the **Time step** element:

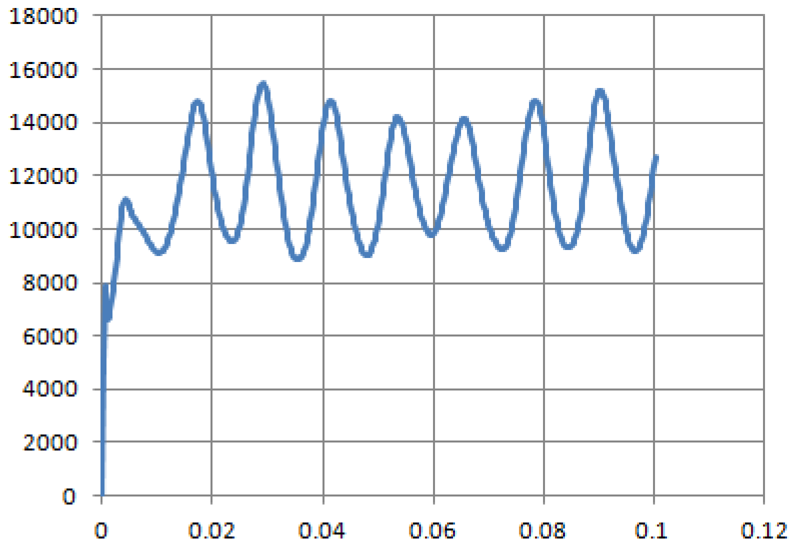
Method	**= Via CFL number**	
Convective CFL	**= 100**	
Slide CFL	**= 1**	
Max step	**= 0.001**	[s]

5.2.3.8 Visualization

To view the dynamics of the solution during the computation, prior to computation, specify visualization of:

1. **Pressure variation on inlet**
2. **Pressure distribution** in the plane of rotation of the blades

5.2.3.8.1 Pressure variation on inlet



- Create a **Supergroup** on the BC **Inlet** using the command **Create supergroup > In Preprocessor** from the context menu.
- Create **Characteristics** on this **Supergroup**.
- In the **Properties** window of the appropriate **Characteristics** object, which locates in the **Preprocessor** tab of the project tree, specify:

Variable	
Variable	**= Pressure**

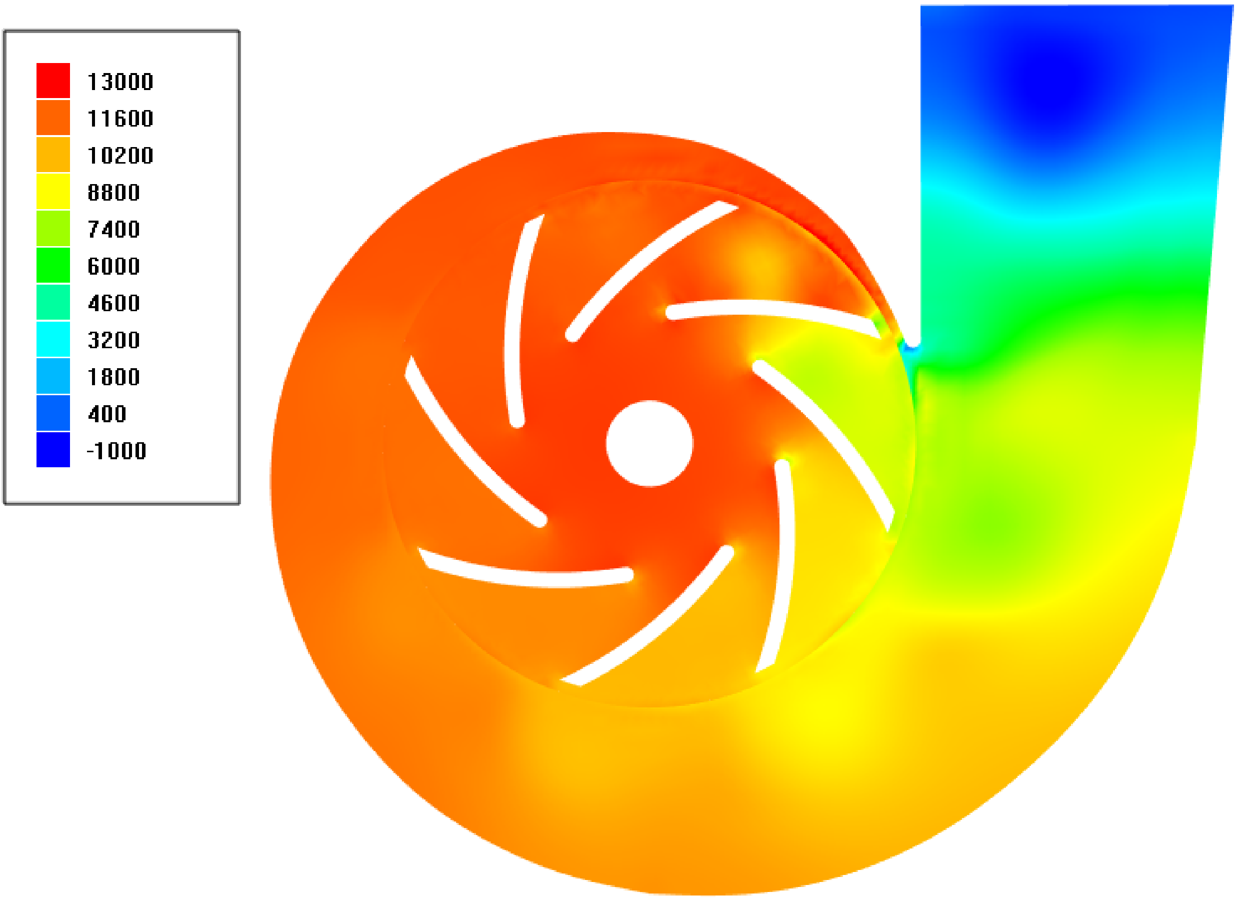
- In the **Properties** window of the appropriate **Characteristics** object, which locates in the **Postprocessor** tab of the project tree, specify:

Save to file

Type = **Automatic**

- When the computation is finished, open the `glo`-file recorded, which is recorded from the **Characteristics** and make a plot of **Avg** depending on **Time**.

5.2.3.8.2 Pressure distribution



- In the **Properties** window of **Plane #0** specify:

Object

Reference point

X = **0**

Y = **0**

Z = **0.01**

Normal

X = **0**

Y = **0**

Z = **1**

(to direct a **Plane**'s normal along the axis **Z**, you can also click the **Operations >** button in the **Properties** window of the **Plane**)

- Create a layer **Color contours** on **Plane #0**.
- In the **Properties** window of the layer specify:

Variable

Variable **Pressure**

Value range

	Mode	**= Manual**	
	Max	**= 13000**	
	Min	**= -1000**	

Palette

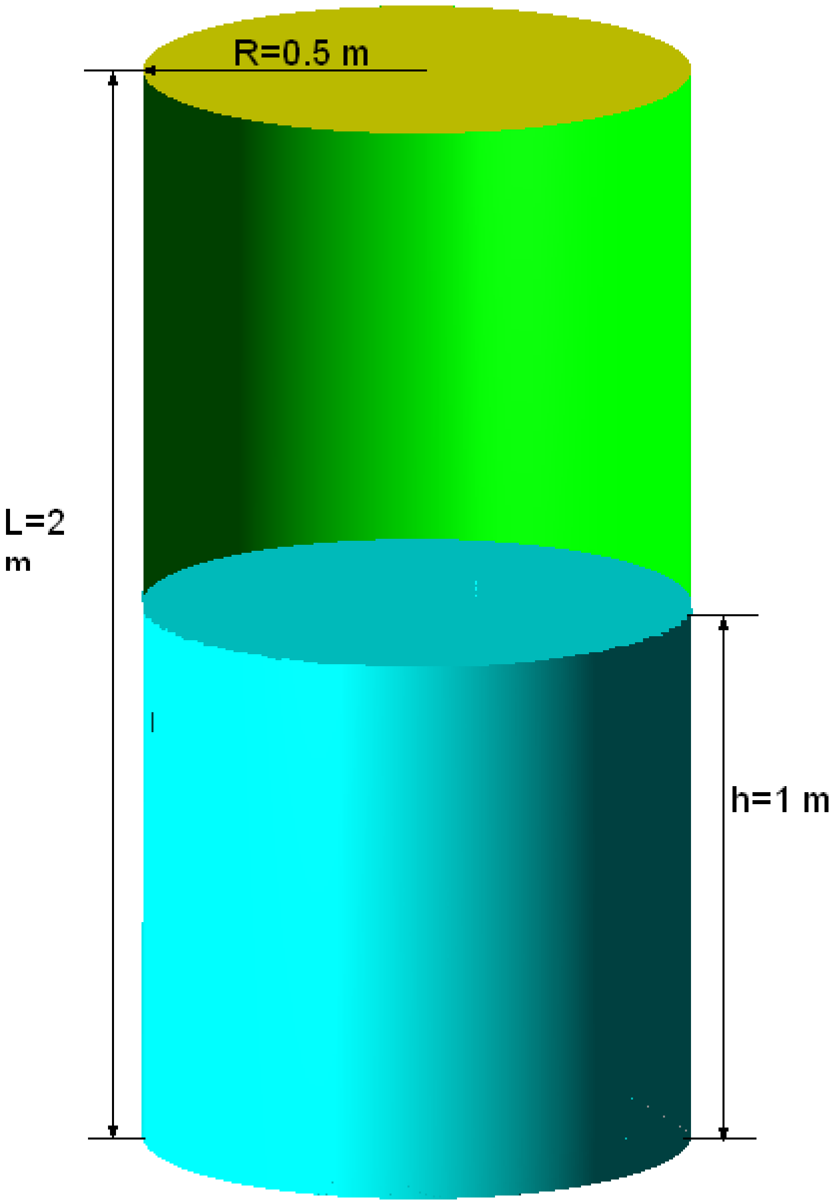
Appearance		
	Enabled	**= Yes**
	Style	**= Style 3**
	Color	**= Black**

Note:

In the current version of the program inaccuracies might appear in the displaying of layers when solving problems with sliding surfaces.

5.2.4 Rotating tank

This example illustrates simulation of a rotating tank, which is half-filled with some liquid.



Parameters of the problem setting

Dimensions:

Height of the tank	L	= 2	[m]
Radius of the tank	R	= 0.5	[m]

Parameters of the wall

Speed of rotation	ω	= 10	[radian s^{-1}]

Fluid parameters:

Density	ρ	= 1000	[kg m^{-3}]
Viscosity	μ	= 100	[kg $m^{-1}s^{-1}$]
Level of the liquid	h	= 1	[m]

Geometry: `Bak.STL`

Project: `Bak`

5.2.4.1 Physical model

In the **Properties** window of the element **General settings** specify:

Gravity vector

X	**= 0**	[m s^{-2}]
Y	**=-9.8**	[m s^{-2}]
Z	**=0**	[m s^{-2}]

In the folder **Substances**:

- Create **Substance #0**.
- Specify the following properties of **Substance #0**:

Aggregative state **= Liquid**

Molar mass

Value	**=0.018**	[kg $mole^{-1}$]

Density

Value	**= 1000**	[kg m^{-3}]

Viscosity

Value	**= 100**	[kg $m^{-1}s^{-1}$]

Specific heat

Value	**= 4217**	[J kg^{-1} K^{-1}]

In the folder **Phases**:

- Create a continuous **Phase #0**.
- In **Phase #0** add **Substance #0** into the folder **Substances**.
- Specify in properties of the folder **Phase #0 > Physical processes**:

Motion **= Navier-Stokes model**

- Create a continuous **Phase #1**.

In the folder **Models**:

- Create **Model #0**.
- Add phases **Phase #0** and**Phase #1** into subfolder **Model #0 > Phases**.

5.2.4.2 Rotation

Create a **Local coordinate system**:

- From the context menu of the folder **Local coordinate systems**, select **Create**.

Specify a **Rotation**:

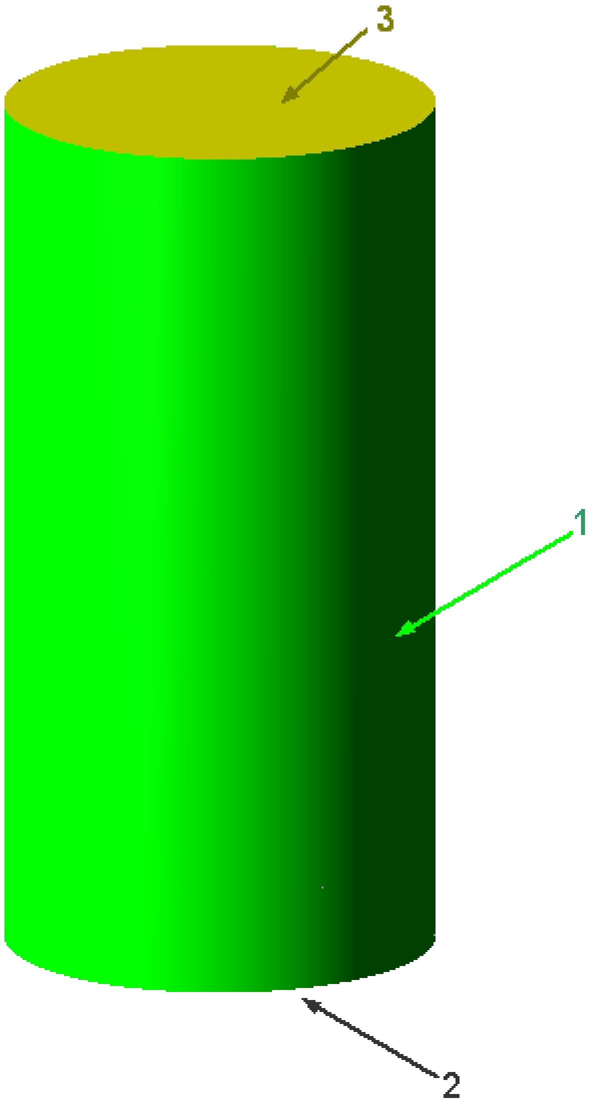
- In the folder **Local CS #0 > Rotation**, create **Rotation #0**.
- In the **Properties** window of **Rotation #0**, specify:

Speed	**= 10**	[radian s^{-1}]
Direction		
X	**= 0**	
Y	**= 1**	
Z	**= 0**	

5.2.4.3 Boundary conditions

In the **Properties** window of **SubRegion #0**, specify:

Model = Model #0



Specify the following boundary conditions:

Boundaries 1, 2

Type	**= Wall**	
Local CS	**= Local CS #0**	
Rotation	**= Rotation #0**	
Variables		
Velocity (Phase #0)	**= No slip**	
VOF(Phase #0)	**=Symmetry**	

Boundary 3

Type	**= Free Outlet**	
Variables		
Velocity (Phase #0)	**= Pressure**	
Value	**= 0**	[Pa]
VOF (Phase #0)	**= Zero gradient**	

5.2.4.4 Initial conditions

Specify the initial volume of the liquid:

- In properties of the element **Model #0 > Init. data > Init. data #0**, specify:

VOF

Value	**= 1**

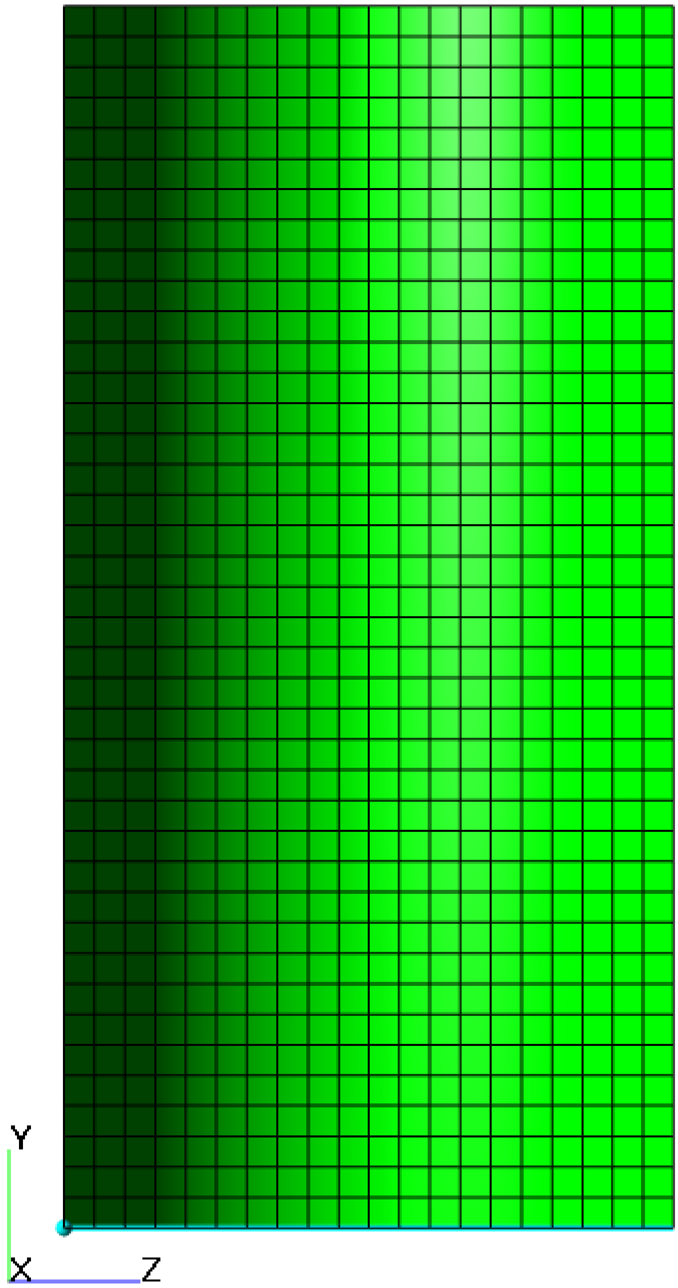
- In the folder **Objects** create **Box #0**.
- In the **Properties** window of **Box #0** specify:

Location		
Reference point		
X	**= 0**	[m]
Y	**= 0.5**	[m]
Z	**= 0**	[m]
Size		
X	**= 1**	[m]
Y	**= 0.99**	[m]
Z	**= 1**	[m]

- In **SubRegion #0**, in the **Properties** window of the element **Initial conditions > Init. condition #0**, specify:

Object	**= Box #0**
Init. data	**= Init. data #0**

5.2.4.5 Initial grid



Specify in the **Properties** window of the **Initial grid**:

nX **= 20**

nY **= 40**

nZ **= 20**

In the **Properties** window of the **Initial grid** click **Apply**.

5.2.4.6 Adaptation of the computational grid

Specify the adaptation of the computational grid on the boundary condition, which corresponds to the side wall of the tank:

- From the context menu of the folder **Computational grid > Adaptation**, select **Create**, so **Adaptation #0** will be created.

- From the context menu of the element **Adaptation #0 > Objects** select the command **Add/Remove Boundary Conditions** and, in the **Select boundary conditions** dialog box, move the boundary condition, which corresponds to the side wall of the tank, from the pane **Not selected** to the pane **Selected**, and then click **OK**.
- In the **Properties** window of the element **Adaptation #0** specify:

Enabled	**= Yes**
Max level N	**= 2**
Layers	
Layers for Level N	**= 4**
Layers for Level N-1	**= 4**

5.2.4.7 Parameters of calculation

In the **Solver** tab, in properties of the **Time step** element, specify:

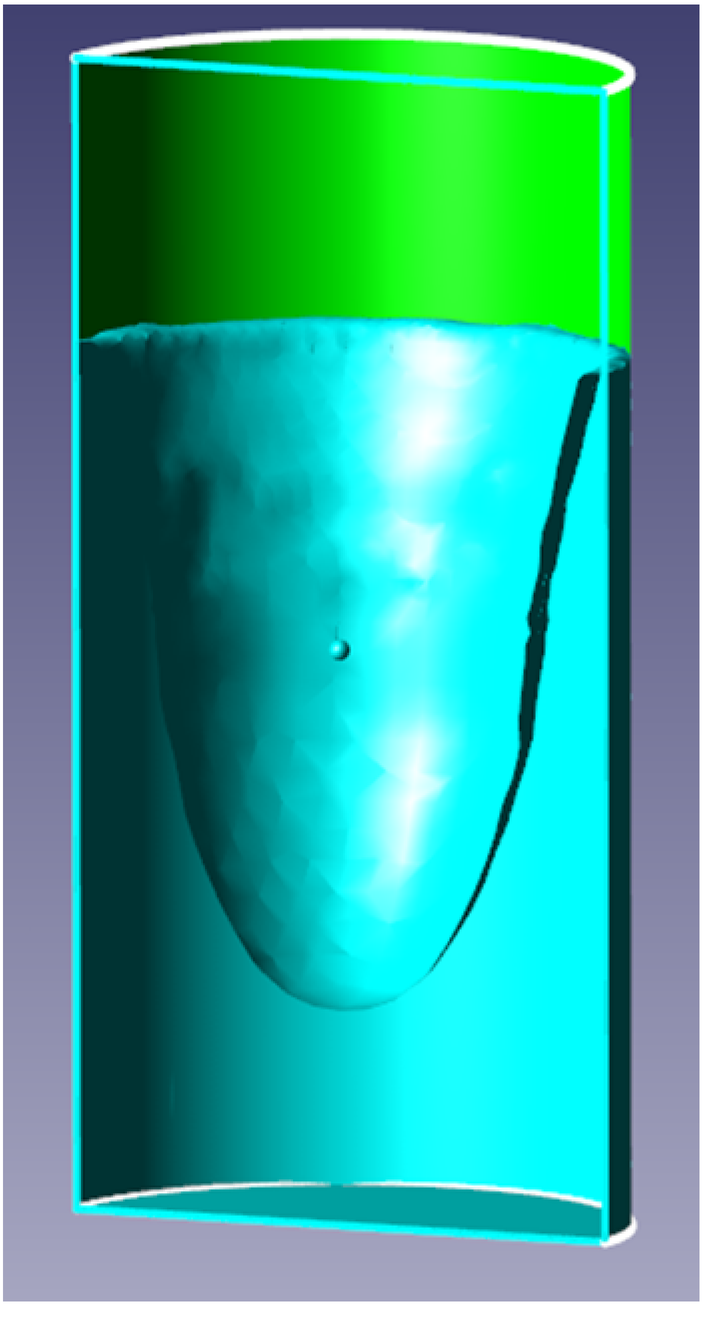
Method	**= Via CFL number**	
Surface CFL	**= 1**	
Max step	**= 1**	[s]

5.2.4.8 Visualization

To view the dynamics of the solution during the computation, specify visualization of the liquid surface layer before the start of computation.

5.2.4.8.1 Surface of the liquid

Visualization at the step number 2000:



- Create a **VOF** layer on the **Computational space**.
- In properties of this layer, specify:

Clipped		**= Yes**
Appearance		
Fill		
	Color	**= Aqua**

- In properties of **Plane #0** specify **Clipping object = Yes**.

Notes:

If you wish to hide some layers or objects displayed in the **View** window, use the **Hide** command from their context menus.

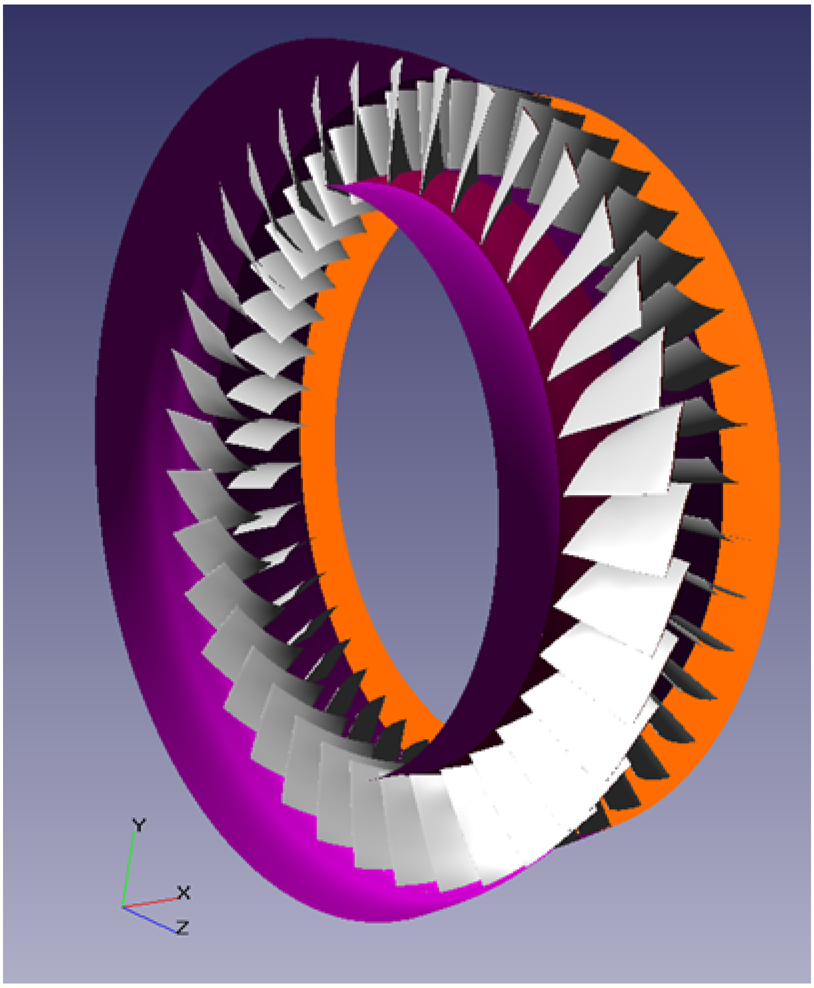
5.2.5 Sector of axial compressor

This example illustrates use of a sector-sliding setting for simulation of a transonic axial compressor. The flow will be simulated in only one sector of the compressor, with use of sliding and periodic boundary conditions.

The simulated compressor is split into sectors, and boundaries between adjacent sectors are set as periodic boundary conditions.

This simulation uses the techniques of sliding meshes.

This class of problems often requires researches of interference between stationary components and rotating blades of the compressor. The techniques of sliding meshes allows the mesh, which is connected to the blade ring of the rotor, to rotate relatively the stationary mesh, which is connected to the stator's blades. Absence of a procedure of permanent regeneration of a mesh around rotating rotor's blades increases the precision of the solution and reduces the required calculation time. This exercise is based on the well-known *NASA stage37* compressor.



Parameters of the problem setting

Number of rotor's blades		= 36	
Number of stator's blades		= 46	
Size:			
Radial clearance		= 0.365×10^{-3}	[m]
Input parameters			
Angular speed of rotor's rotation	ω	= 1800	[radian s^{-1}]
Working medium		= air	
Reference pressure	P_{ref}	= 101325	[Pa]

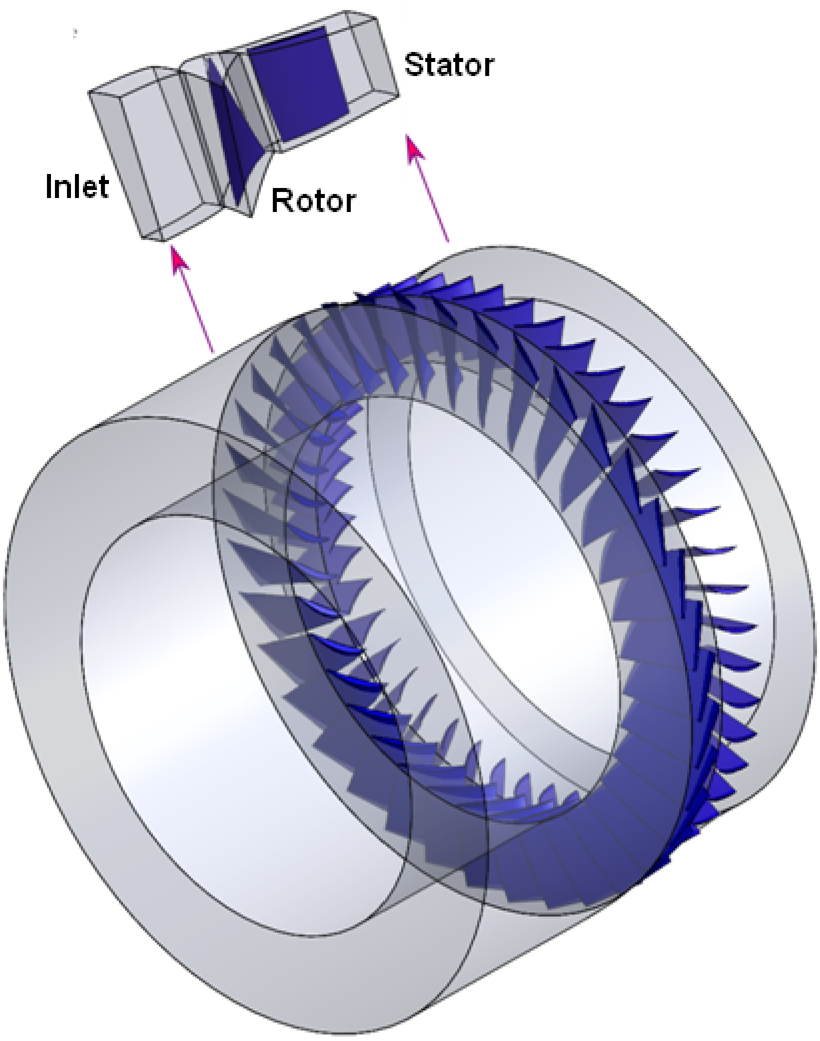
Reference temperature	T_{ref}	= 283	[K]
Geometry:	**`NASA_stage37.wrl`**		
Project:	**NASA_stage37**		

5.2.5.1 Making geometry of the computational domain

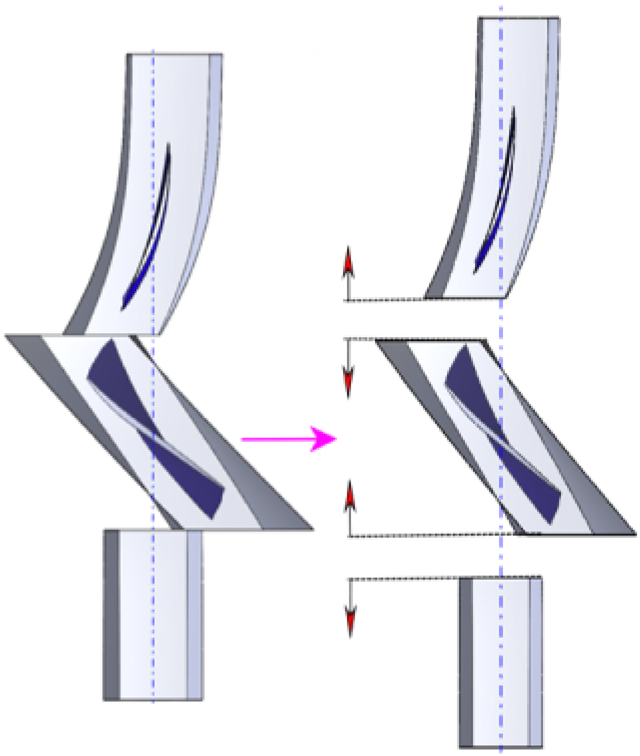
In simulations of problems with rotating bodies, when the flow is assumed to be axially symmetric relatively to the axis of rotation, to reduce the calculation time, it is desired to select a sector containing the geometry of the shroud and of one compressor's blade, and simulate the flow in only this sector.

For this, it is necessary to modify the compressor's geometry in the CAD software. The specified sector should pack into the full circle (360 degrees) an integer number of times. In the *NASA stage37* axial compressor the rotor's wheel has 36 blades, and the stator's blade contains 46 blades. So the sector, which contains a rotor's blade, should be multiple of 360/36 = 10 degrees, and the sector, which contains a stator's blade, should be multiple of 360/46 = 7.826 degrees. In this exercise we used the minimal possible sector angles that are multiple of these values.

The inlet part can be combined with the rotor's sector or, as in our exercise (to demonstrate possible functionality), be specified as a separate sector with angle as of 7.5 degrees.



Surfaces of all subregions must be closed and not coincide. To provide this, it is necessary, in the CAD software, to move apart the subregions, obtained after splitting into sectors the common initial geometry of the assembly.



Moving the subregions apart can be done by the following methods:

- by moving the subregions apart along their axis of rotation (this is done so in our exercise, as shown on the illustration above)
- by turning the subregions around their axis of rotation so their conjugated surfaces will not coincide

5.2.5.2 Physical model

Load into **Pre-Postprocessor** the previously prepared geometry of the project, **`NASA_stage37.wrl`**.

After loading the geometry, *FlowVision* will automatically recognize three subregions.

In the folder **Substances**:

- Create **Substance #0**.
- Load **Substance #0** from the **Standard** substance database:
  - From the context menu of **Substance #0** select **Load from SD > Standard**.
  - In the new window **Load from database**, select:

Substances	**= Air**
Phases	**= Gas (equilibrium)**

In the folder **Phases**:

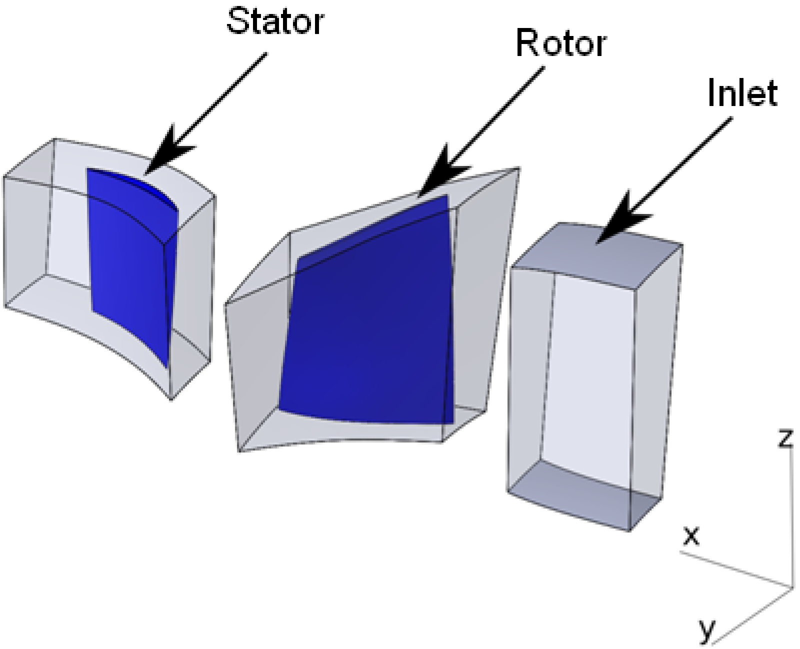
- Create a continuous **Phase #0**.
- Add the **Air_Gas (equilibrium)** substance into the folder **Phase #0 > Substances**.
- In properties of the folder **Phase #0 > Physical processes**, specify:

Heat transfer	**= Heat transfer via H**
Motion	**= Navier-Stokes model**
Turbulence	**= KES**

In the folder **Models**:

- Create **Model #0**.
- Add **Phase #0** into subfolder **Model #0 > Phases**.

Rename the recognized **Subregions** as **Inlet**, **Rotor**, and **Stator**. Specify **Model = Model #0** in properties of each of these **Subregions**.



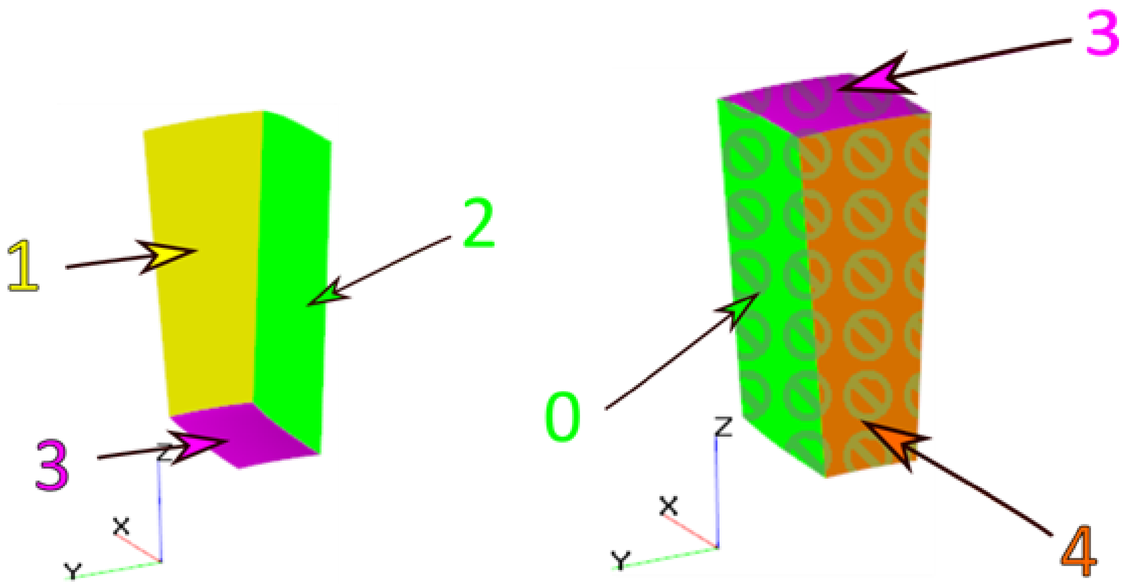
In the folder **Models > Model #0 > Init. data > Init. data #0** specify:

Velocity (Phase #0)

X	**= 150**	[m s^{-1}]

5.2.5.3 Boundary conditions

Boundary conditions of the subregion "Inlet"



In the subregion **Inlet** specify the following boundary conditions:

Boundary 0

Name	**= Periodic1**
Type	**= Connected**

Boundary 1

Name	**= Inlet-Rotor**
Type	**= Connected**

Boundary 2

Name	**= Periodic2**
Type	**= Connected**

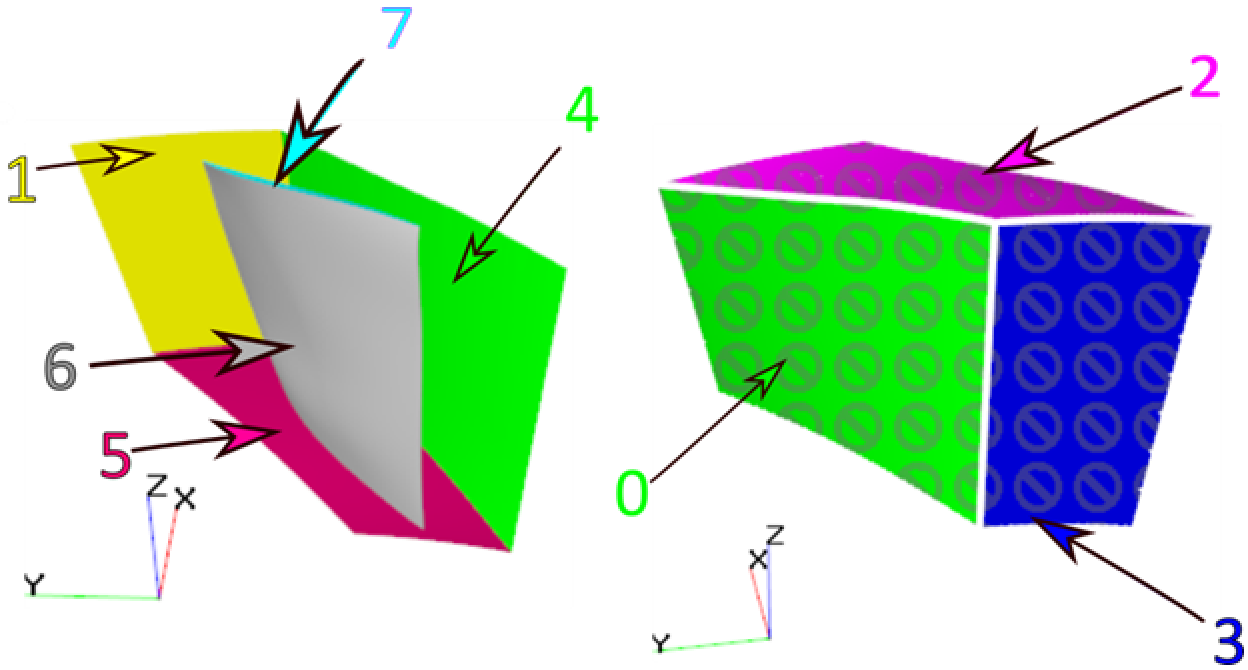
Boundary 3

Name	**= Shroud**
Type	**= Wall**
Variables	
Temperature (Phase #0)	**= Zero gradient**
Velocity (Phase #0)	**= Logarithm law**
TurbEnergy (Phase #0)	**= Value in cell near wall**
TurbDissipation (Phase #0)	**= Value in cell near wall**

Boundary 4

Name	**= Inlet**
Type	**= Inlet/Outlet**
Variables	
Temperature (Phase #0)	**= Total temperature**
Value	**= 0** [K]
Velocity (Phase #0)	**= Total pressure**
Value	**= 0** [Pa]
TurbEnergy (Phase #0)	**= Pulsations**
Value	**= 0**
TurbDissipation (Phase #0)	**= Turbulent scale**
Value	**= 0**

Boundary conditions of the subregion "Rotor"



In the subregion **Rotor** specify the following boundary conditions:

Boundary 0

Name **= Periodic1**

Type **= Connected**

Boundary 1

Name **= Rotor-Stator**

Type **= Connected**

Boundary 2

Name **= Shroud**

Type **= Wall**

Variables

Temperature (Phase #0) **= Zero gradient**

Velocity (Phase #0) **= Logarithm law**

TurbEnergy (Phase #0) **= Value in cell near wall**

TurbDissipation (Phase #0) **= Value in cell near wall**

Boundary 3

Name **= Inlet-Rotor**

Type **= Connected**

Boundary 4

Name **= Periodic2**

Type **= Connected**

Boundary 5

Name **= Hub**

Type **= Wall**

Variables

Temperature (Phase #0) = Zero gradient
Velocity (Phase #0) = Logarithm law
TurbEnergy (Phase #0) = Value in cell near wall
TurbDissipation (Phase #0) = Value in cell near wall

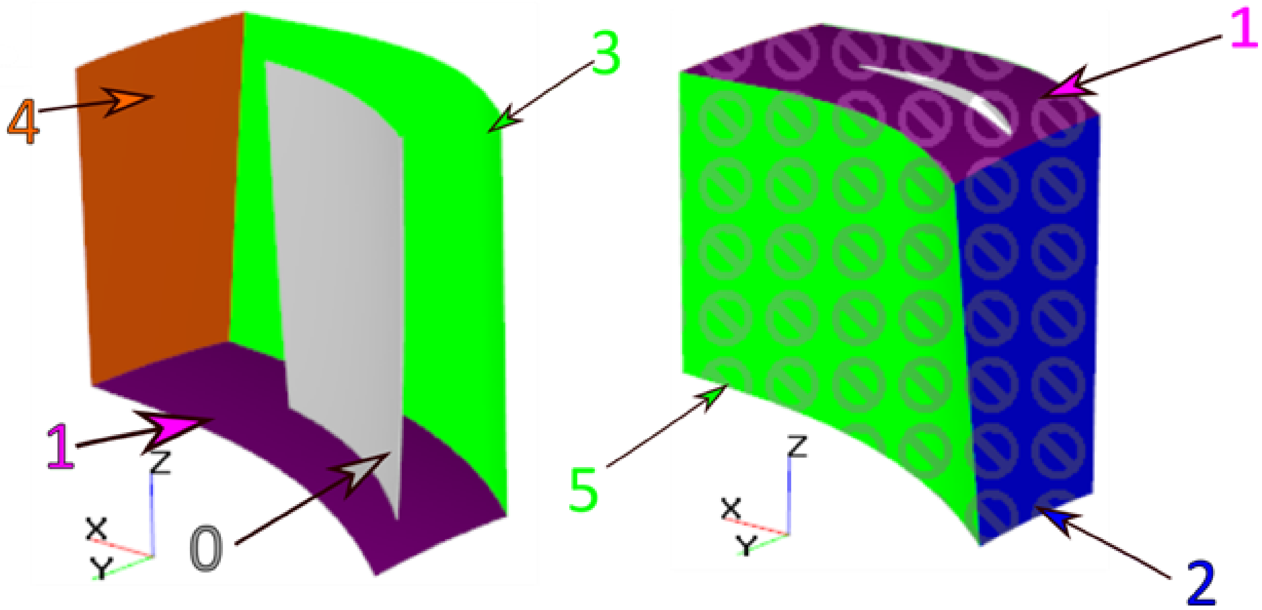
Boundary 6

Name = **Blade**
Type = **Wall**
Variables
Temperature (Phase #0) = **Zero gradient**
Velocity (Phase #0) = **Logarithm law**
TurbEnergy (Phase #0) = **Value in cell near wall**
TurbDissipation (Phase #0) = **Value in cell near wall**

Boundary 7

Name = **Blade tip**
Type = **Wall**
Variables
Temperature (Phase #0) = **Zero gradient**
Velocity (Phase #0) = **Logarithm law**
TurbEnergy (Phase #0) = **Value in cell near wall**
TurbDissipation (Phase #0) = **Value in cell near wall**

Boundary conditions of the subregion "Stator"



In the subregion **Stator** specify the following boundary conditions:

Boundary 0

Name = **Blade**
Type = **Wall**
Variables

Temperature (Phase #0) **= Zero gradient**
Velocity (Phase #0) **= Logarithm law**
TurbEnergy (Phase #0) **= Value in cell near wall**
TurbDissipation (Phase #0) **= Value in cell near wall**

Boundary 1

Name **= Shroud**
Type **= Wall**
Variables
Temperature (Phase #0) **= Zero gradient**
Velocity (Phase #0) **= Logarithm law**
TurbEnergy (Phase #0) **= Value in cell near wall**
TurbDissipation (Phase #0) **= Value in cell near wall**

Boundary 2

Name **= Rotor-Stator**
Type **= Connected**

Boundary 3

Name **= Periodic2**
Type **= Connected**

Boundary 4

Name **= Outlet**
Type **= Free Outlet**

Boundary 5

Name **= Periodic1**
Type **= Connected**

5.2.5.4 Specifying the Rotation and binding Subregions

To use the technique of sliding meshes, it is necessary to bind **Subregions**. For this, you have to create a **Local coordinate system** having a **Rotation** element.

This element **Rotation** has to be assigned to those interface boundary conditions, from which the working medium (gas) will flow from one **Subregion** to another.

In this exercise, the computational area consists of three **Subregions** and has two interface surfaces, so the binder conditions will be created for:

- two pairs of **Boundary conditions**, connecting the **Subregions**
- *and* three pairs of periodic **Boundary conditions**

Create a **Local coordinate system**:

- From the context menu of the folder **Local coordinate systems**, select **Create.**

Specify a **Rotation**:

- In the folder **Local CS #0 > Rotation** create **Rotation #0.**
- In the **Properties** window of **Rotation #0**, specify:

Speed **= 1800** [radian s^{-1}]
Direction
X **= 1**
Y **= 0**

Z **= 0**

In the subregion **Inlet**, specify:

- In the **Properties** window of the boundary condition **Inlet-Rotor**, specify:

  Local CS **= Local CS #0**

  Rotation **= Rotation #0**

In the subregion **Rotor**, specify:

- In the **Properties** window of the boundary condition **Inlet-Rotor**, specify:

  Local CS **= Local CS #0**

  Rotation **= Rotation #0**

- In the **Properties** window of the boundary condition **Rotor-Stator**, specify:

  Local CS **= Local CS #0**

  Rotation **= Rotation #0**

In the subregion **Stator**, specify:

- In the **Properties** window of the boundary condition **Rotor-Stator**, specify:

  Local CS **= Local CS #0**

  Rotation **= Rotation #0**

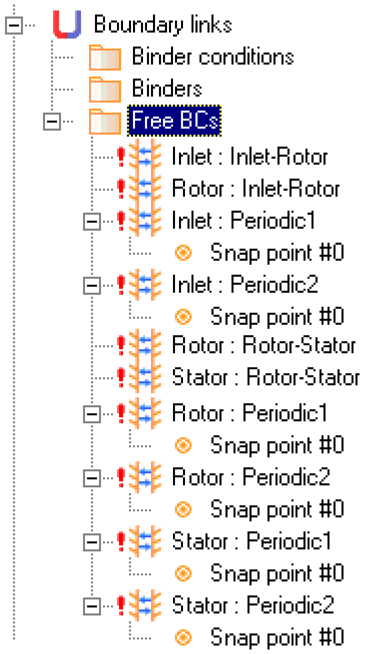
As it already has explained in the exercise Sector of a rotor, when boundary conditions for periodic surfaces are being bound, it is necessary to specify **Snap points**. Let's specify a pair of **Snap points** per each binder, which is used to bind periodic surfaces.

For this, in the folder **Free BCs**, from context menus of both **Boundary conditions** that are to be bound, select the command **Create snap point**. Then, in the **Properties** windows of the both snap points, if necessary, specify their coordinates or move the **Snap points** using icon buttons with arrows (, , ,) in such way, that, at binding the periodic surfaces, these **Snap points** will correctly match each other.

> Correct matching of **Snap points** might require their movement.
>
> Buttons and (thick arrows) in the **Properties** windows of a **Snap point** move the **Snap point** along the contour of the periodic surface to the next sharp bend of the contour. It is recommended to use these buttons when the contour has evident sharp bends.
>
> Buttons and (thin arrows) in the **Properties** windows of a **Snap point** move the **Snap point** along the contour of the periodic surface to the next vertex of the polygon, which approximates the curvilinear contour. This buttons move the **Snap point** relatively slowly and without selecting positions of sharp bends; this is less convenient and increases the risk of errors.
>
> Also you can enter coordinates of the **Snap point** and then click **Apply**.

In the folder **Free BCs**, create **Snap points** for the following **Boundary conditions**:

- **Inlet: Periodic1**
- **Inlet: Periodic2**
- **Rotor: Periodic1**
- **Rotor: Periodic2**
- **Stator: Periodic1**
- **Stator: Periodic2**



> You don't have to create snap points for binding sliding surfaces, and, after a **Local CS** (local coordinate system) and **Rotation** are specified on such surfaces, it will be impossible to specify snap points on these surfaces.

Create **Binders**:

- From the context menu of the folder **Binders**, select **Create all**.

Create **Binder condition #0**:

- From the context menu of the folder **Binder conditions**, select **Create**.
- In the **Create binder condition** dialog box, which opens, specify:

Connection type	**= Periodic surface**
1st model	**= Model #0**
2nd model	**= Model #0**

Specify the matching between the **Binders** and the **Binder condition**:

- From the context menu of the folder **Binder conditions > Binder condition #0 > Binders**, select **Add/Remove** and click **Add All** button in the **Select binders** dialog box, which opens.

Create **Binder condition #1**:

- From the context menu of the folder **Binder conditions**, select **Create**.
- In the **Create binder condition** dialog box, which opens, specify:

Connection type	**= Sliding surface**
1st model	**= Model #0**
2nd model	**= Model #0**

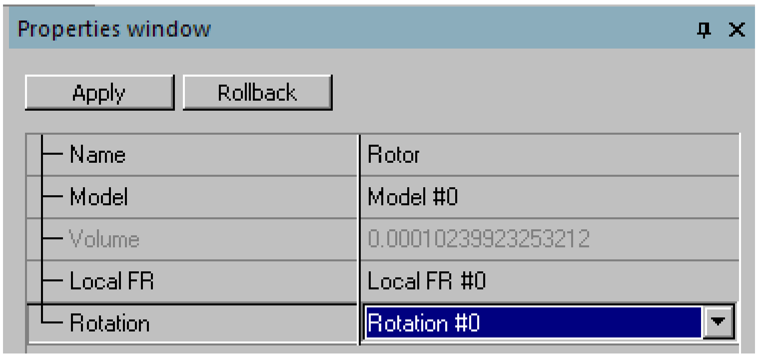
Specify the matching between the **Binders** and the **Binder condition**:

- From the context menu of the folder **Binder conditions > Binder condition #1 > Binders**, select **Add/Remove** and click **Add All** button in the **Select binders** dialog box, which opens.

5.2.5.5 Specifying the rotor's rotation

In the **Properties** window of the subregion **Rotor**, specify:

Local CS	**= Local CS #0**
Rotation	**= Rotation #0**



Also specify **Local CS = Local CS #0** and **Rotation = Rotation #0** in properties of boundary conditions in the subregion **Rotor**:

- In the **Properties** window of the boundary condition **Blade**, specify:

  Local CS **= Local CS #0**

  Rotation **= Rotation #0**

- In the **Properties** window of the boundary condition **Hub**, specify:

  Local CS **= Local CS #0**

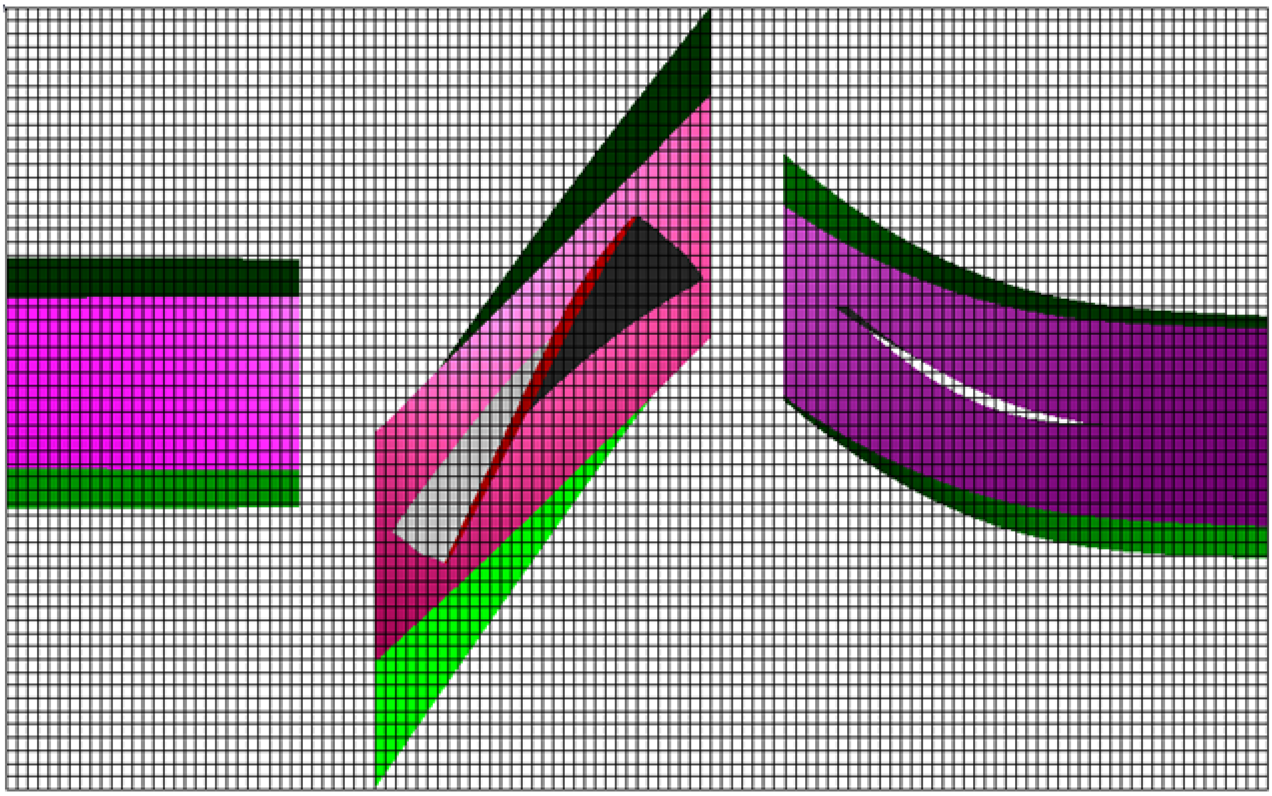
  Rotation **= Rotation #0**

- In the **Properties** window of the boundary condition **Blade tip**, specify:

  Local CS **= Local CS #0**

  Rotation **= Rotation #0**

5.2.5.6 Initial grid



Specify in the **Properties** window of the **Initial grid**:

nX **= 120**

nY **= 60**

nZ **= 50**

In the **Properties** window of the **Initial grid** click **Apply**.

5.2.5.7 Adaptation of the computational grid

Specify the adaptation in the subregion **Rotor**:

- Create the element **Computational grid > Adaptation > Adaptation #0**.
- From the context menu of the element **Adaptation #0 > Subregions** select **Add/Remove** and, in the **Select Subregions** dialog box, place **Rotor** into **Selected**, and click **OK**.
- From the context menu of the element **Adaptation #0 > Objects** select the command **Add/Remove Boundary Conditions** and, in the **Select boundary conditions** dialog box, place **Rotor : Blade** into **Selected** and click **OK**.
- In the **Properties** window of the element **Adaptation #0** specify:

Enabled **= Yes**

Max level N **= 1**

Layers

Layers for Level N **= 2**

- Create the element **Computational grid > Adaptation > Adaptation #1**.
- From the context menu of the element **Adaptation #1 > Subregions** select **Add/Remove** and, in the **Select Subregions** dialog box, place **Rotor** into **Selected**, and click **OK**.
- From the context menu of the element **Adaptation #1 > Objects** select the command **Add/Remove Boundary Conditions** and, in the **Select boundary conditions** dialog box, place **Rotor : Blade tip** into **Selected** and click **OK**.
- In the **Properties** window of the element **Adaptation #1** specify:

Enabled **= Yes**

Max level N **= 2**

Layers

Layers for Level N **= 2**

Layers for Level N-1 **= 2**

Specify the adaptation in the subregion **Stator**:

- Create the element **Computational grid > Adaptation > Adaptation #2**.
- From the context menu of the element **Adaptation #2 > Subregions** select **Add/Remove** and, in the **Select Subregions** dialog box, place **Stator** into **Selected**, and click **OK**.
- From the context menu of the element **Adaptation #2 > Objects** select the command **Add/Remove Boundary Conditions** and, in the **Select boundary conditions** dialog box, place **Stator : Blade** into **Selected** and click **OK**.
- In the **Properties** window of the element **Adaptation #2** specify:

Enabled **= Yes**

Max level N **= 1**

Layers

Layers for Level N **= 2**

Note: This project uses a simplified computational grid. To obtain good results, use a grid with higher resolution.

5.2.5.8 Parameters of calculation

In the **Solver** tab of the project tree, specify:

- In the **Properties** window of the element **Time step**, specify:

Method	**= Via CFL number**
Convective CFL	**= 100**
Max step	**= 0.001** [s]

- In the **Properties** window of the element **Advanced settings**, specify:

Numerical method

Advection scheme	**= 1st order scheme**
Type of scheme	**= Implicit**

Sliding surfaces

Method *)	**= Frozen Rotor**

*) Problems in sector-sliding settings are recommended to be simulated using the **Frozen Rotor** method.

- In the **Properties** window of the element **Limiters > Limiters for calculation > Phase Limiters > Phase #0**, specify:

Limiter

Density, min.	**= 0.001**	[kg m^{-3}]
Temperature abs, min.	**= 100**	[K]
Temperature abs, max.	**= 500**	[K]
Velocity, max.	**= 10e+4**	[m s^{-1}]
Pressure abs, min.	**= 100**	[Pa]
Pressure abs, max.	**= 10e+9**	[Pa]

5.2.5.9 Visualization

To view the dynamics of the solution during the computation, prior to computation, specify visualization of:

1. **Mass flow variation**
2. **Mach Number distribution** on a cylindrical surface, which goes through **Inlet**, **Rotor** and **Stator**

5.2.5.9.1 Mass flow variation

This problem is a quasi-steady-state one. To find the convergence of the solution, it is necessary to watch values of algebraic residuals.

Also this class of problems has another character parameter, the mass flow. To watch the mass flow, lets create a **Characteristics** in the **Preprocessor** tab of the project tree.

In the project tree, in the **Preprocessor** tab, open the context menu of the object **Subregions > Inlet > Boundary conditions > Inlet** and, selecting there the command **Create supergroup > In Preprocessor**, create a **Supergroup** on the boundary condition **Inlet**.

Create **Characteristics #0** on the new just created **Supergroup**.

In the **Properties** window of **Characteristics #0**, specify:

Variable

Variable	**= Pressure**

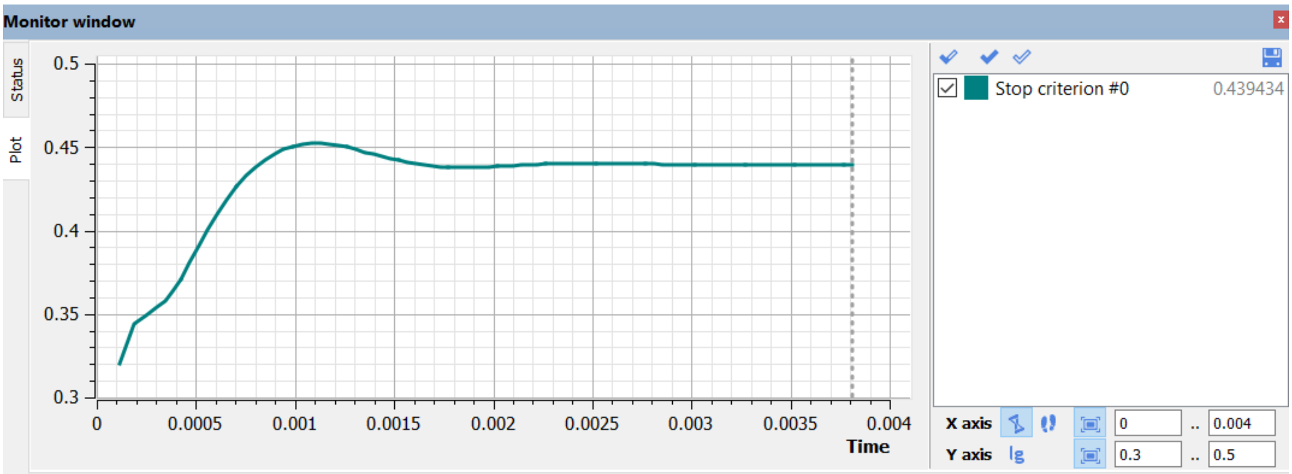
Create a **Stop criterion**:

In the project tree, in the **Solver** tab, in the folder **Stopping conditions > User values**, create **Stop criterion #0**.

In the **Properties** window of **Stop criterion #0**, specify:

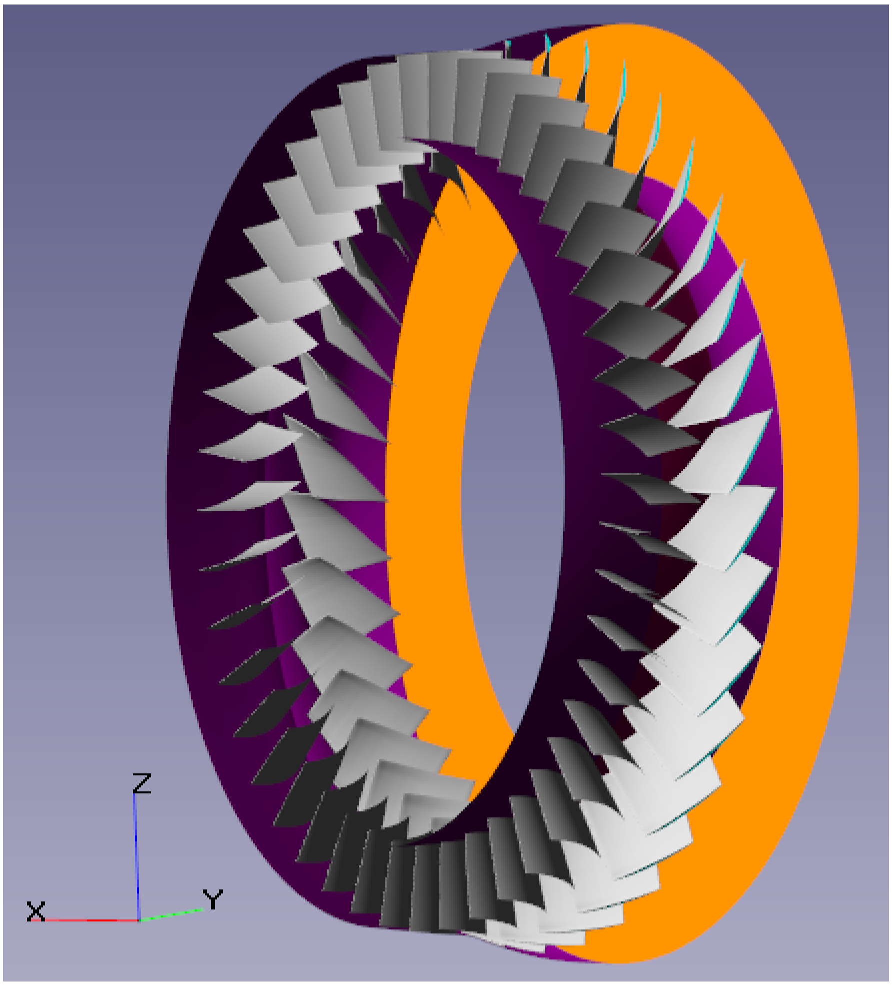
Level	**= 1e-4**
Object	**= Characteristics #0 (Supergroup on "Inlet")**
Variable	**= Mass flow+**

Stop criterion #0, which you have just created, you can watch variation of the variable **Mass flow+** immediately in **Pre-Postprocessor**, in the **Plot** tab of the **Monitor** window:

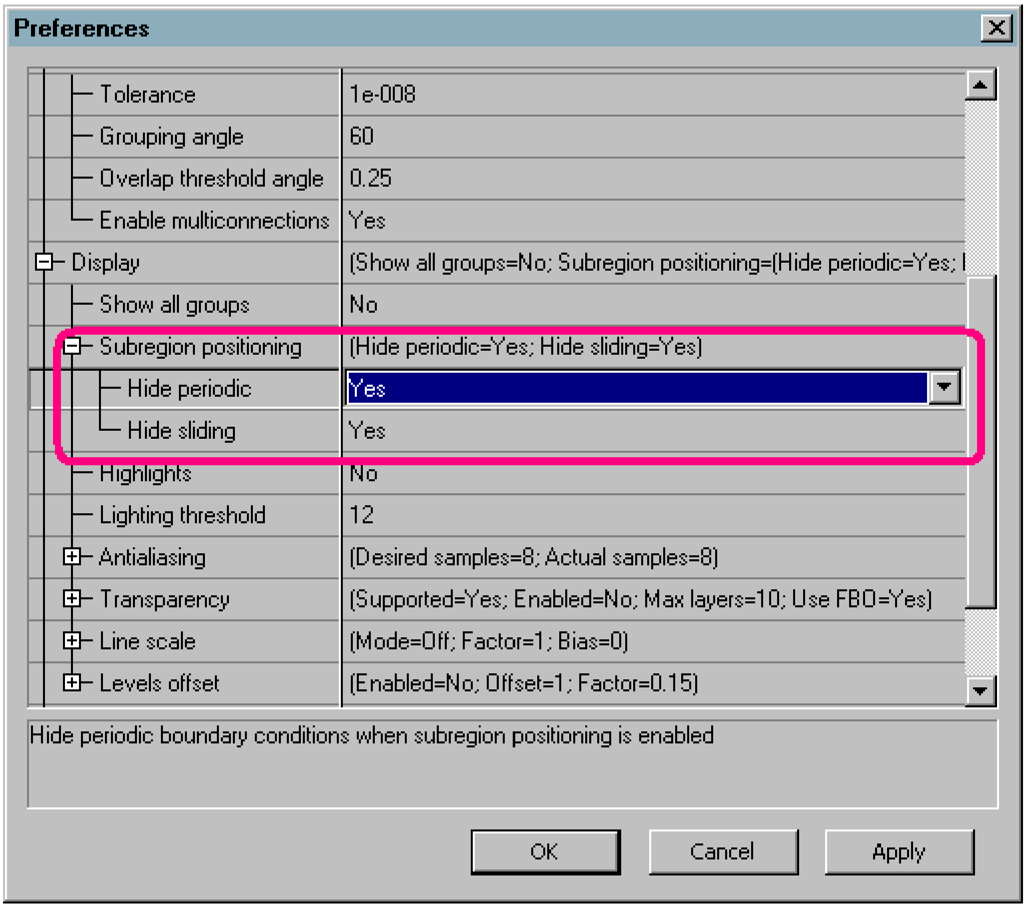


5.2.5.9.2 Mach Number distribution

To view the whole rotor, stator, and circular inlet of the compressor, click the button (**Enable/disable duplication and overlapping of subdomains up to the complete model supplied with sector-sliding**) in the **Rendering** toolbar. Images of **Subregions** will be multiplied (repeatedly copied and placed around the compressor's axis of rotation):



Displaying or hiding periodic and/or slicing boundary conditions in the image, formed when you click the  button, can be tuned by parameters **Display > Subregion positioning > ...** in the **Preferences** window, which is opened by the **File > Preferences** command from the main menu:



From the main menu select the command **File > Preferences** and in the **Preferences** window, which opens, specify:

Display > Subregion positioning > Hide periodic **= Yes**

Display > Subregion positioning > Hide sliding **= Yes**

Note: Making multiple images of **Subregions** will require additional resources of RAM memory on the **Solver**'s side and video RAM memory on the side of the client computer.

In the project tree, in the **Postprocessor** tab, create a geometric object **Cone/cylinder**.

In the **Properties** window of the just created object **Cone/cylinder #0**, specify the following parameters:

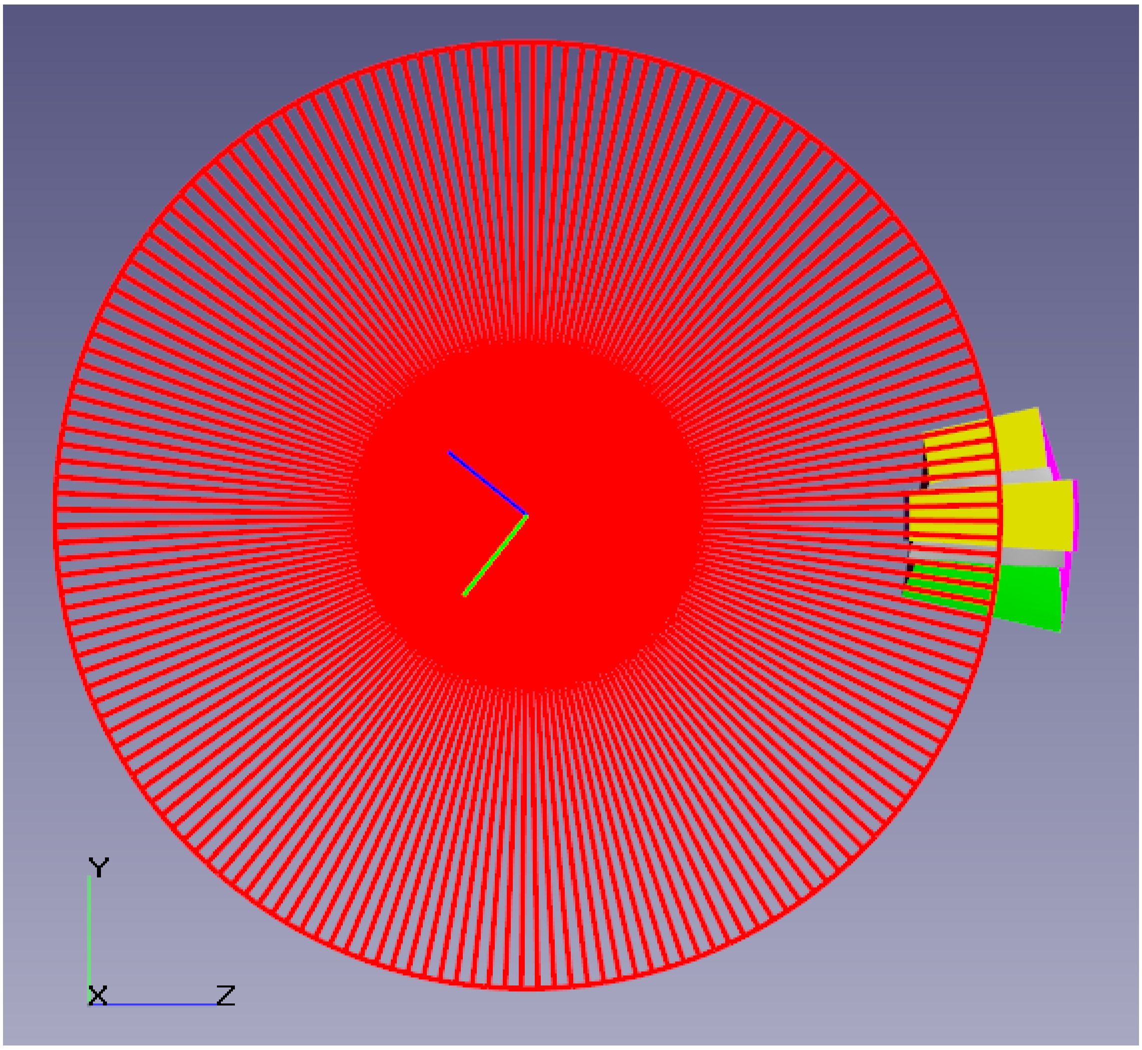
Object

Location > Reference point

X	**= -0.05**	[m]
Y	**= 0**	
Z	**= 0**	

Parameters

Height	**= 0.15**	[m]
Radius 1	**= 0.22**	
Radius 2	**= 0.22**	
Base ratio	**= 1**	



On the object **Cone/cylinder #0**, create a **Color contours** layer.

In the **Properties** window of the just created layer **Color contours #0 (Cone/cylinder #0)**, specify:

Parts

Select	**= Selected surfaces**
Surfaces > Lateral surface	**= Yes**
Surfaces > Bottom base	**= No**
Surfaces > Top base	**= No**

Variable

Variable	**= MachNumber**

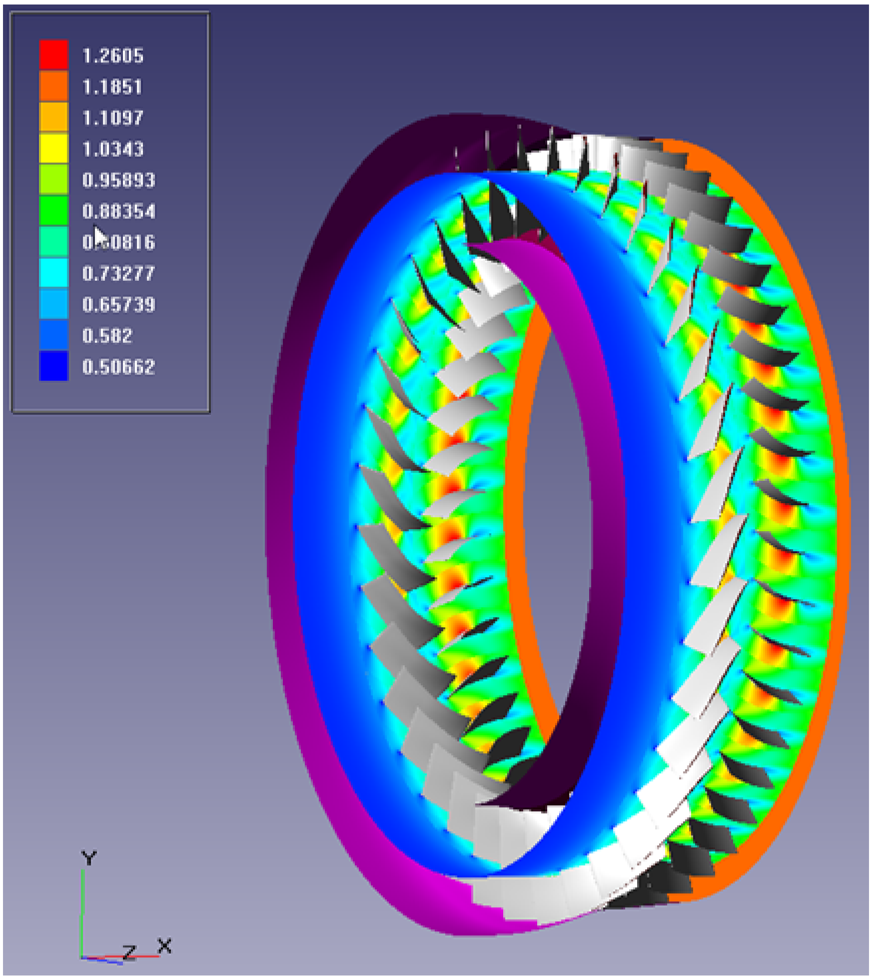
Value range

Mode	**= Local**

Palette

Appearance > Enabled	**= Yes**

During the project's computation, the visualization of the Mach Number distribution will be formed:


1.2605
1.1851
1.1097
1.0343
0.95893
0.88354
0.73277
0.65739
0.582
0.50662
Y
Z
X

5.3 Moving bodies

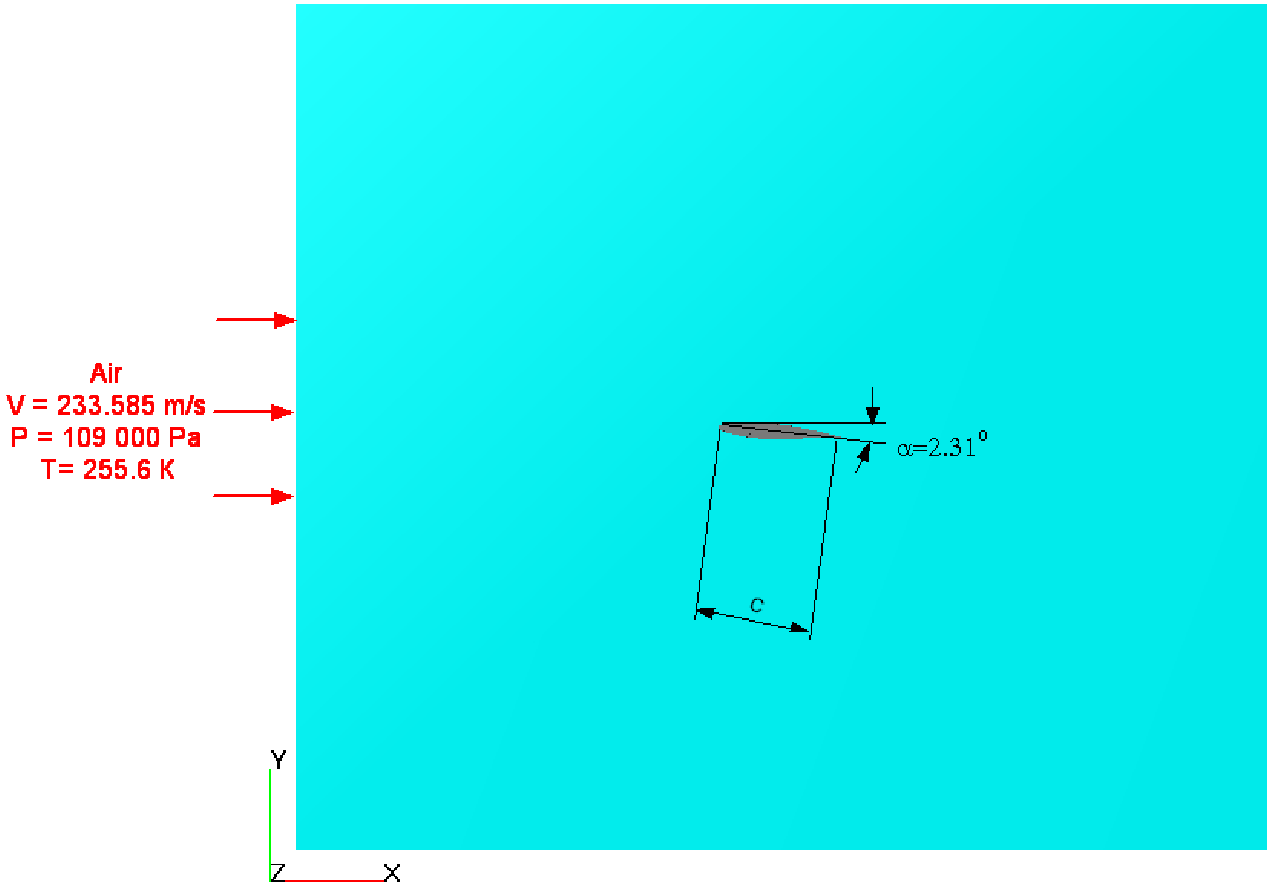
FlowVision allows changing the initial position of individual elements of the geometry and simulating of their translational or rotational movement. This is supported by Moving body modifiers. The geometry of a moving body is loaded from files, which are similar to the files with the basic geometry.

In order to simulate a task with moving bodies, it is necessary to:

- Create geometric models of the **Moving bodies** in *FlowVision* or using a third-party CAD software.
- Based on these geometric models, create **Imported objects**.
- Create **Moving bodies**.
- Disable updates of those **Moving bodies**, which will not move during the computation.
- Specify a law of motion for those **Moving bodies**, which will move during the computation.
- Specify appropriate boundary conditions on the surfaces of **Moving bodies**.
- It is advisable to use the value of the time step about 1x**Surface CFL**.
- If **Moving bodies** are moving in incompressible fluid under the action of forces from the fluid, and oscillations occur, it is recommended to specify (in properties of the **Substance**) the **Density**, which depends on **Pressure** and/or **Temperature** (for example, by specifying partial derivatives of **Density** with respect to **Pressure** and/or to **Temperature**, **dRho/dP** and **dRho/dT**). If the oscillations do not disappear, it is recommended to enable the **Artificial compressibility** in properties of the physical process **Motion** and specify values of damping coefficients in properties of the **Moving bodies** (parameters **Translation > Damping parameter** and **Rotation > Damping parameter**).

5.3.1 Transonic flow around an airfoil

This example illustrates a transonic flow around the *RAE 2822* airfoil with the angle of attack of 2.31 degrees.



Parameters of the problem setting

Dimensions:

Chord length:	c	= 0.3	[m]
Dimensions of the computational domain		13.8 × 12 × 0.1	[m × m × m]
Angle of attack:	α	= 2.31 °	

Substance: = Air

Inlet parameters:

Static pressure:	P	= 109 000	[Pa]
Static temperature:	T	= 255.6	[K]
Velocity on inlet:	V_{inl}	=233,585	[$m\ s^{-1}$]
Mach number:	M	= 0.73	
Reynolds number:	Re	= 6.5×10^6	

Geometry: **`RAE_2822_Domain.wrl`**

Project: **`RAE_2822`**

Note:

Computation of this project might require significant computing resources and a long time.

5.3.1.1 Physical model

In the **Properties** window of **Region**, specify:

Tolerance **= 1e-10**

In the **Properties** window of the element **General settings** specify:

Reference values

Temperature	**= 255.6**	[K]
Pressure	**= 109000**	[Pa]

In the folder **Substances**:

- Create **Substance #0**.
- Load **Substance #0** from the **Standard** substance database:

Substances **= Air**

Phases **= Gas (equilibrium)**

In the folder **Phases**:

- Create a continuous **Phase #0**.
- Add the **Air_Gas (equilibrium)** substance into the folder **Phase #0 > Substances**.
- Specify in properties of the folder **Physical processes**:

Motion **= Navier-Stokes model**

Heat transfer **= Heat transfer via H**

Turbulence **= SA**

In the folder **Models**:

- Create **Model #0**.

- Add **Phase #0** into subfolder **Model #0 > Phases**.
- In the folder **Init. data #0**, specify:

Velocity

X	**= 233.585**	[m s^{-1}]
Y	**= 0**	[m s^{-1}]
Z	**= 0**	[m s^{-1}]

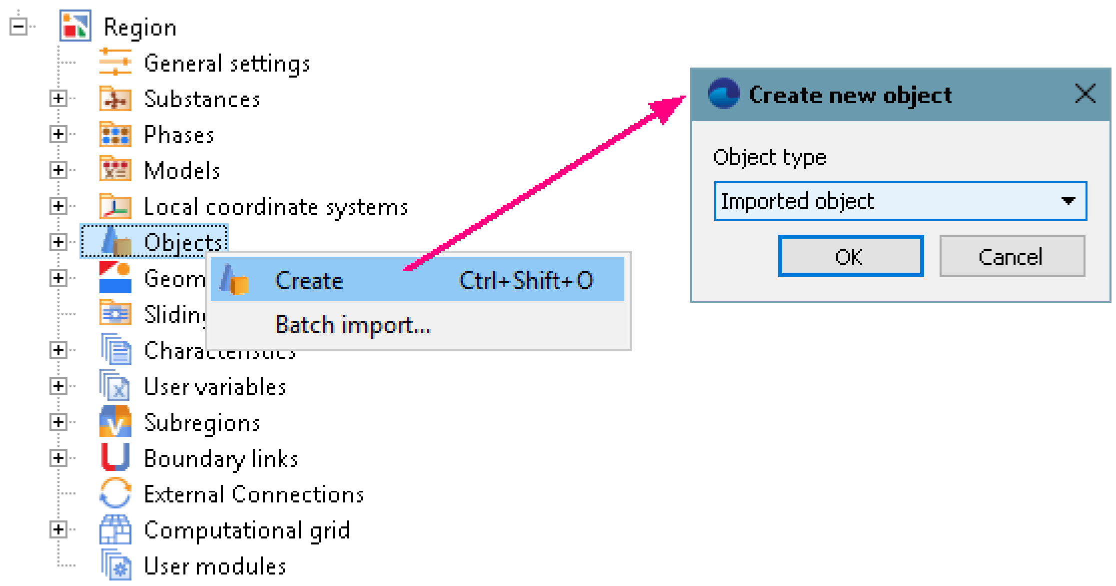
5.3.1.2 Moving body

In order to be able to define the attack angle from the interface, it is necessary to specify the airfoil profile as a **Moving body**.

Creation of a moving body consists of the following steps:
- creation the geometry of the **Moving body**
- creation a **Moving body** modifier

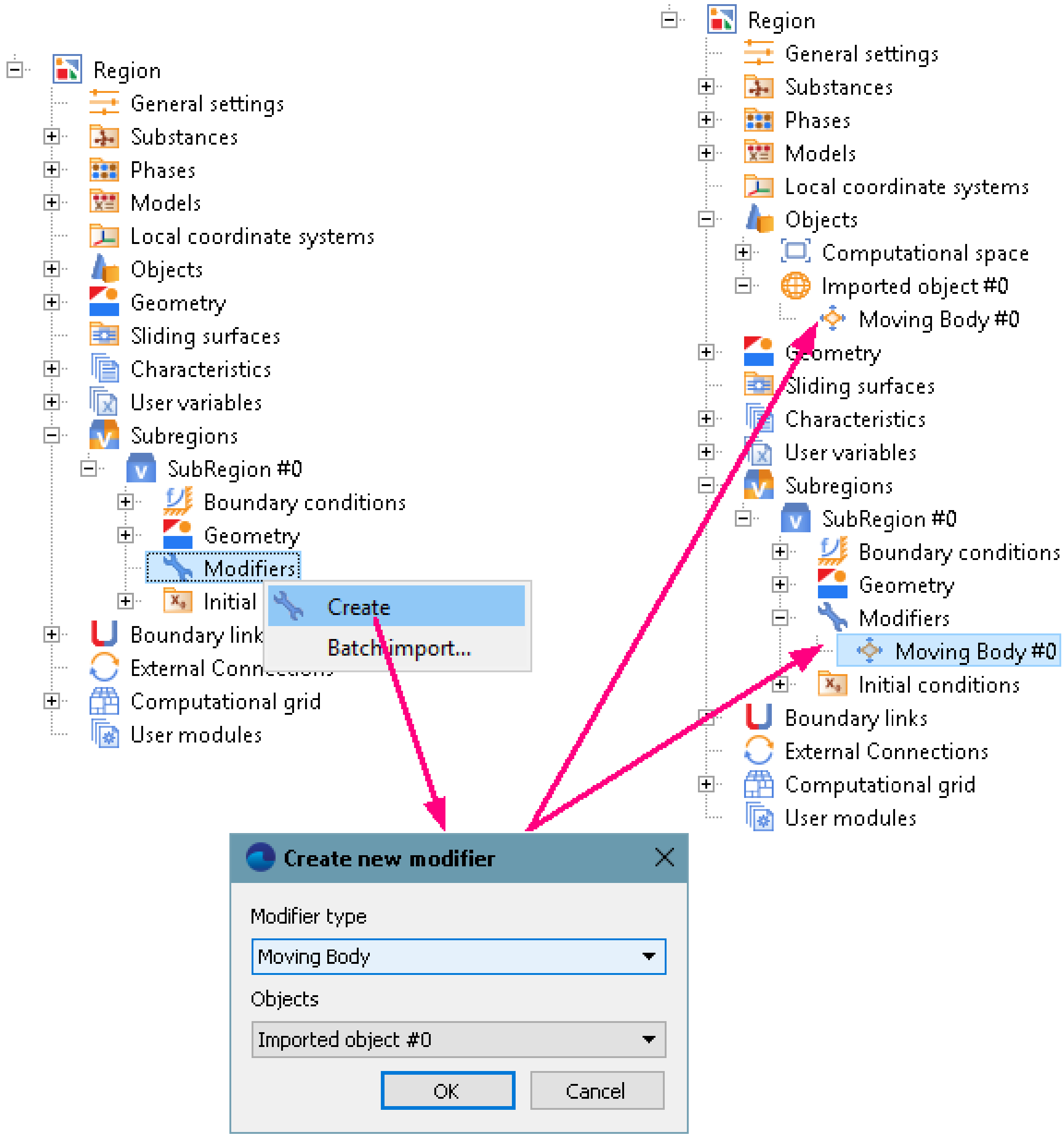
Geometry of a **Moving body** can be created:
- in a third-party geometric modeling software and then be loaded into *FlowVision* as an **Imported object**
- in *FlowVision*

In this example, the geometry of the **Moving body** is loaded in *FlowVision* from a file. To load the geometry of the **Moving body**, do the following steps:
- Select **Create** from the context menu of the folder **Objects**.
- Specify **Object type** = **Imported object**.
- Download the geometry of the moving body from the file `RAE_2822_Airfoil.STL`.



To assign the status of a moving body to the **Imported object**, do the following:
- In the **Properties** window of **SubRegion #0**, specify:

  Model = Model #0
- In the folder **Modifiers**, create a new **Moving body** modifier based on an **Imported object #0**.



- In the **Properties** window of the **Moving body**, specify:

  Initial position

  Axis X

X	**= 0.999187**
Y	**= -0.04030**
Z	**= 0**

- In the **Properties** window of the **Moving body**, click **Operations >**  **(Place to initial position)**.
- In the **Properties** window of the **Moving body**, specify:

  Update

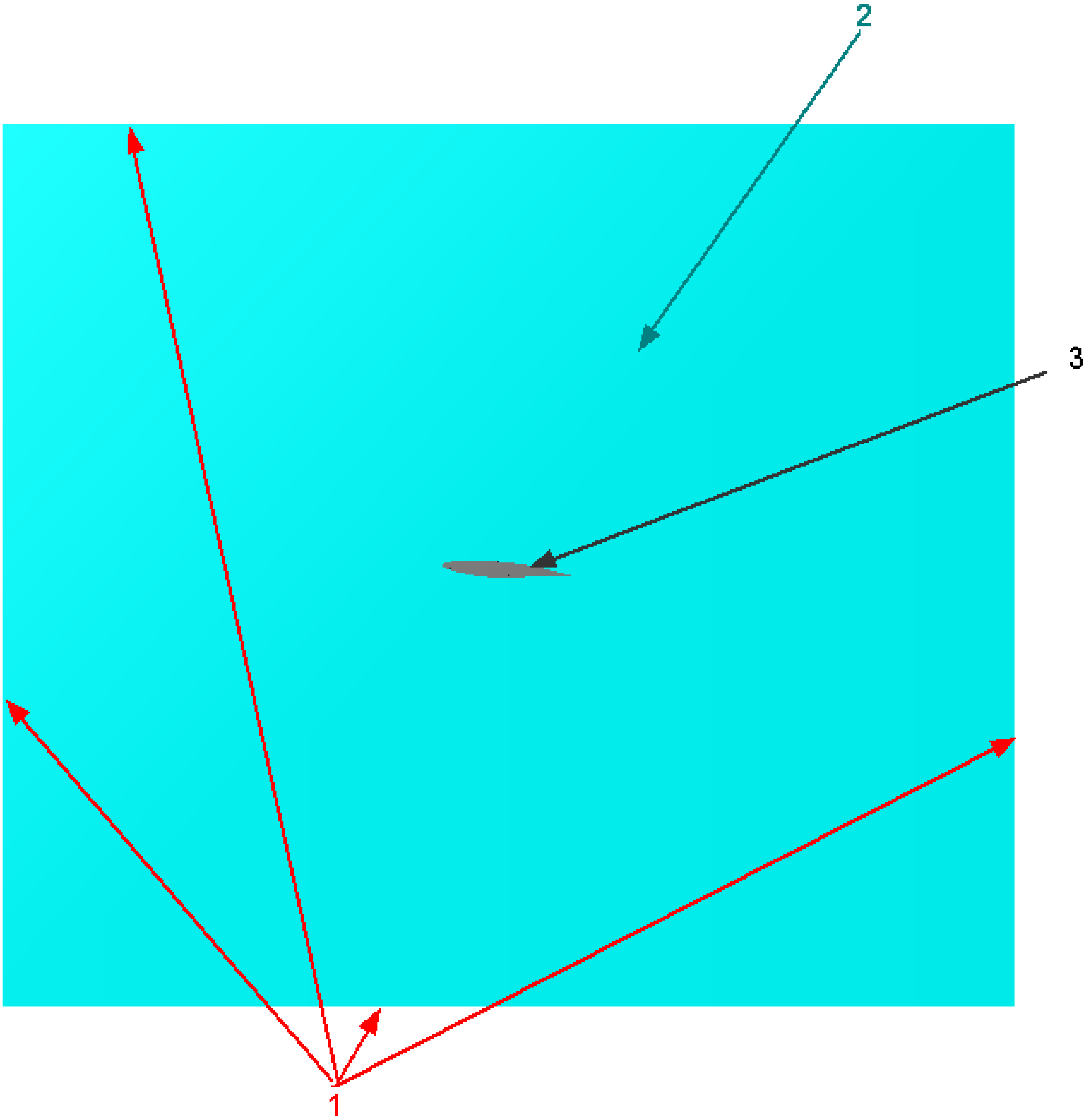
  Type **= Disabled**

Notes:

1. After you disable updates of a **Moving body**, its **Mass Properties**, parameters of **Translational** and **Rotation movement**, **Initial position**, **Limitation** becomes unavailable for editing. So the **Initial position** is to be specified *before* you disable the **Update**.

2. When update of a **Moving body** is enabled, then rebuilding of the grid is done on each iteration. This operation is resource consuming and wastes CPU time. Therefore, in simulations where the **Moving body** actually does not move, it is recommended to disable its **Update**.
3. If there are several **Moving bodies** in a simulation, then update of at least one of them will cause update of the others.

5.3.1.3 Boundary conditions



Specify the following boundary conditions:

Boundary 1

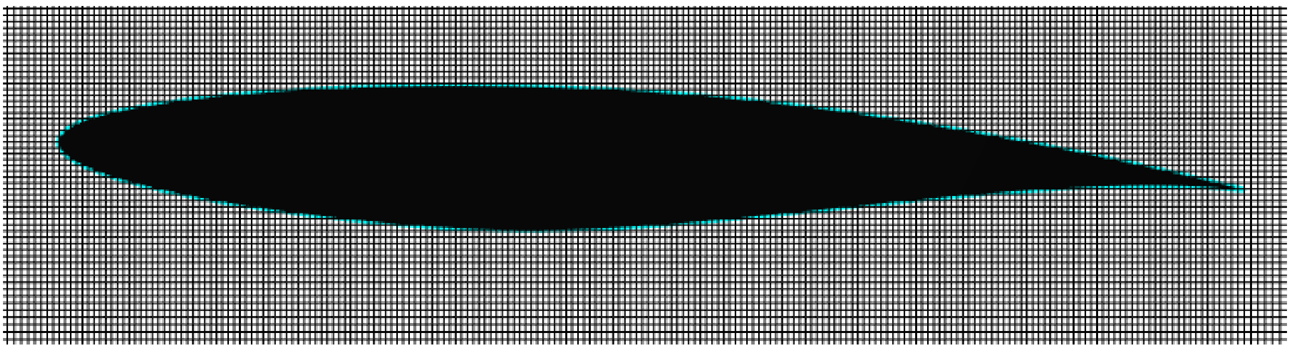
Type	**= Non-reflecting**	
Variables		
Temperature(Phase #0)	**= Non-reflect.**	
Value	**= 0**	[K]
Velocity(Phase #0)	**= Non-reflect.**	
Velocity at inf.		

X = **233.585** [m s^{-1}]
Y
Z
Pressure at inf. = **0** [Pa]
TurbKinViscosity(Phase #0) = **Value**
Value = **0**

Boundary 2
Type = **Symmetry**
Variables
Temperature(Phase #0) = **Symmetry**
Velocity(Phase #0) = **Slip**
TurbKinViscosity(Phase #0) = **Symmetry**

Boundary 3
Type = **Wall**
Variables
Temperature(Phase #0) = **Zero gradient**
Velocity(Phase #0) = **Logarithm law**
TurbKinViscosity(Phase #0) = **Value in cell near wall**

5.3.1.4 Initial grid



In the **Properties** window of the **Initial grid**, click the button  to open the **Initial grid editor**.
Specify in the **Initial grid editor**:

for axis OX

Grid parameters
h_max = **1.5** [m]
h_min = **0.0015** [m]

Insert a reference line with coordinate **x=0** [m].

Specify **Reference line parameters** for the reference line with coordinate **x=-6**:
h = **1.5** [m]
kh+ = **1**

Specify **Reference line parameters** for the reference line with coordinate **x=0**:
h = **0.0015** [m]

kh- **= 0.97**

kh+ **= 0.97**

Specify **Reference line parameters** for the reference line with coordinate **x=7.8**:

h **= 1.5** [m]

kh- **= 0.93**

for axis OY (click the button )

Grid parameters

h_max **= 1.5** [m]

h_min **= 0.0015** [m]

Insert a reference line with coordinate **y=0** [m].

Specify **Reference line parameters** for the reference line with coordinate **y=-6**:

h **= 1.5** [m]

kh+ **= 1.4**

Specify **Reference line parameters** for the reference line with coordinate **y=0**:

h **= 0.0015** [m]

kh- **= 1**

kh+ **= 1**

Specify **Reference line parameters** for the reference line with coordinate **y=6**:

h **= 1.5** [m]

kh- **= 0.6**

Click **OK** to close the **Initial grid editor** with saving the entered data.

In properties of the **Initial grid** specify:

Grid structure = 2D

Plane = XY

In the **Properties** window of the **Initial grid** click **Apply**.

5.3.1.5 Adaptation of the computational grid

In this project the computation is done in two stages:

- Up to 300-th step the computation is done on the coarse initial grid. At step 300 the solution will be a steady-state one and it will form a supersonic area with a pressure surge on the upper surface of the airfoil. Resolution of this area by the grid, details of the flow in the area and location of the pressure surge after the 300-th step will be found out at the second stage of the computation (see below).
- From the 300-th to the 500-th step an **Adaptation by condition** will be applied that will resolve supersonic areas that were formed at the first stage. Also adaptation near the surface of the airfoil and merging the cells that get out of the supersonic areas. At the 500-th step the solution becomes a steady-state one and you can stop the computation.

So the following adaptations of the computational grid are specified in this project:

- a simple **Adaptation** on the surface of the airfoil
- **Adaptation by condition** M > 1. This adaptation will resolve supersonic area at upper side of the airfoil, its maximal level is 2 and the algorithm creates 20 layers for each level outside, so this adaptation will include the pressure surge after the supersonic area.
- an **Adaptation** for merging to the level 0 the cells that previously were split (adapted) but now are getting out of the supersonic area

Specify a simple **Adaptation** on the boundary condition, which is set on the airfoil's surface (boundary 3):

- Create the element **Computational grid > Adaptation > Adaptation #0**.
- Add the boundary condition **Wall #0**, which is set on the airfoil's surface, to the subfolder **Computational grid > Adaptation > Adaptation #0 > Objects**.
- In properties of **Adaptation #0** specify:

Enabled = **Yes**

Max level N = **2**

Layers

Layers for Level N = **10**

Layers for Level N-1 = **10**

Specify an **Adaptation to solution**, which will resolve the supersonic area over the airfoil and the pressure surge after the supersonic area:

- Create the element **Computational grid > Adaptation by condition > Adaptation by condition #0**.
- Add **Computational space** to the subfolder **Computational grid > Adaptation by condition > Adaptation by condition #0 > Objects**.
- In properties of **Adaptation by condition #0** specify:

Enabled = **Yes**

Max level N = **2**

Layers

Layers for Level N = **20**

Layers for Level N-1 = **20**

Conditions

Variable

Variable = **MachNumber**

Range

From = **1**

To = **10000000000**

Specify a simple **Adaptation** to merge cells. This **Adaptation** will undo the grid refinement that was made by **Adaptation by condition #0**, it will restore the initial size of the cells in places, from which the supersonic area got away:

- Create the element **Computational grid > Adaptation > Adaptation #1**.
- Add **Computational space** to the subfolder **Computational grid > Adaptation > Adaptation #1 > Objects**.
- In properties of **Adaptation #1** specify:

Enabled = **Yes**

Max level N = **0**

Split/Merge = **Merge**

The adaptations should be applied after the solution on the **Initial grid** is converged (becomes steady-state), after the 300-th step. Specify this:

- In properties of the *folder* **Computational grid > Adaptation** specify:

Activation

Type = **Inactive**

- In properties of the *folder* **Computational grid > Adaptation by condition** specify:

Activation

Type = **Repetitive by step**

Start in steps = **300**

Duration in steps = **5**

Period in steps = **50**

According to these settings, **Adaptation #0** and **Adaptation #1** will not be activated independently, because activation of their folder **Computational grid > Adaptation** is disabled (**Activation > Type = Inactive** is set there). However, at the moments when **Adaptation by condition #0** is triggered, these simple adaptations will also be triggered, because **Enabled = Yes** is set in their properties.

> For better understanding this tutorial case, start off with creating and running this project without grid adaptations and watch dynamics of the solution.
> And only after this specify adaptations and run the project with them.

5.3.1.6 Parameters of calculation

Specify in the **Solver** tab:

- In the **Properties** window of the **Time step** element specify:

Method	**= Via CFL number**	
Convective CFL	**= 100**	
Max step	**= 0.01**	[s]

- In the **Properties** window of the **Limiters > Limiters for calculation > Phase limiters > Phase #0** element specify:

Limiter

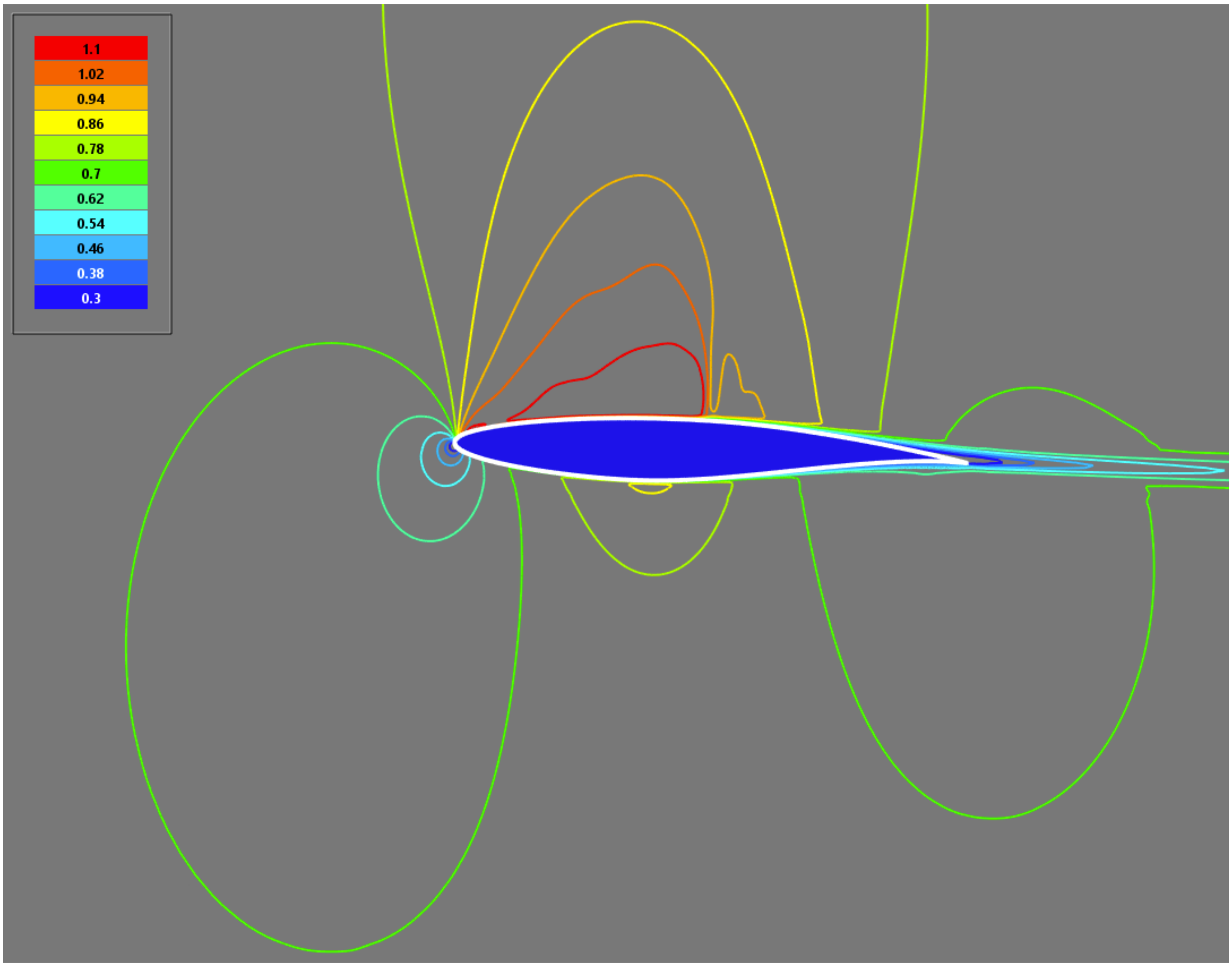
Density, min.	**= 0.001**	[kg m^{-3}]
Velocity, max.	**= 1500**	[m s^{-1}]
Pressure abs, min.	= **100**	[Pa]
Pressure abs, max.	= **1e+8**	[Pa]
Temperature abs, min.	= **100**	[K]
Temperature abs, max.	= **1e+5**	[K]

5.3.1.7 Visualization

To view the dynamics of the solution during the computation, prior to computation, specify visualization of:

1. Mach number distribution in the plane of the flow
2. Cp distribution on the profile's surface

5.3.1.7.1 Mach number distribution



- In the **Properties** window of **Plane #0**, specify:

Object

Normal

X	**= 0**
Y	**= 0**
Z	**= 1**

(to direct a **Plane**'s normal along the axis **Z**, you can also click the **Operations >**  button in the **Properties** window of the **Plane**)

- Create a layer **Color contours** on **Plane #0.**
- In the **Properties** window of the layer, specify:

Variable	
Variable	**MachNumber**
Value range	
Mode	**= Manual**
Max	**= 1.1**
Min	**= 0.3**
Method	**= Isolines**
Palette	
Appearance	
Enabled	**= Yes**

The given illustration shows results of the simulation at the 600th time step.

5.3.1.7.2 Cp distribution

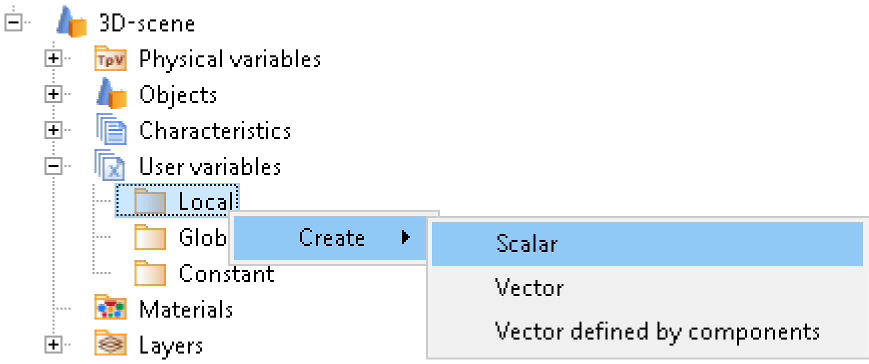
Building the distribution of Cp on the profile's surface consists of the following steps:

1. Creating the variable Cp
2. Creating the "Plot along curve" layer for Cp

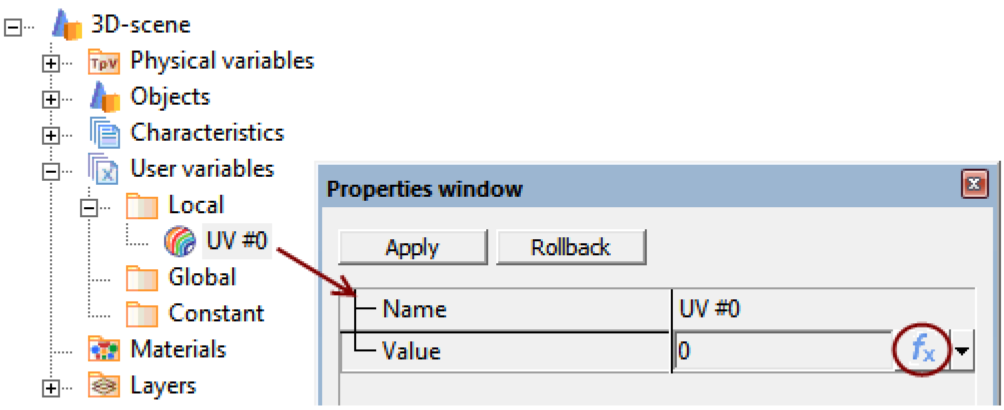
5.3.1.7.2.1 Creating the variable Cp

Create the **Cp** variable in **Postprocessor**:

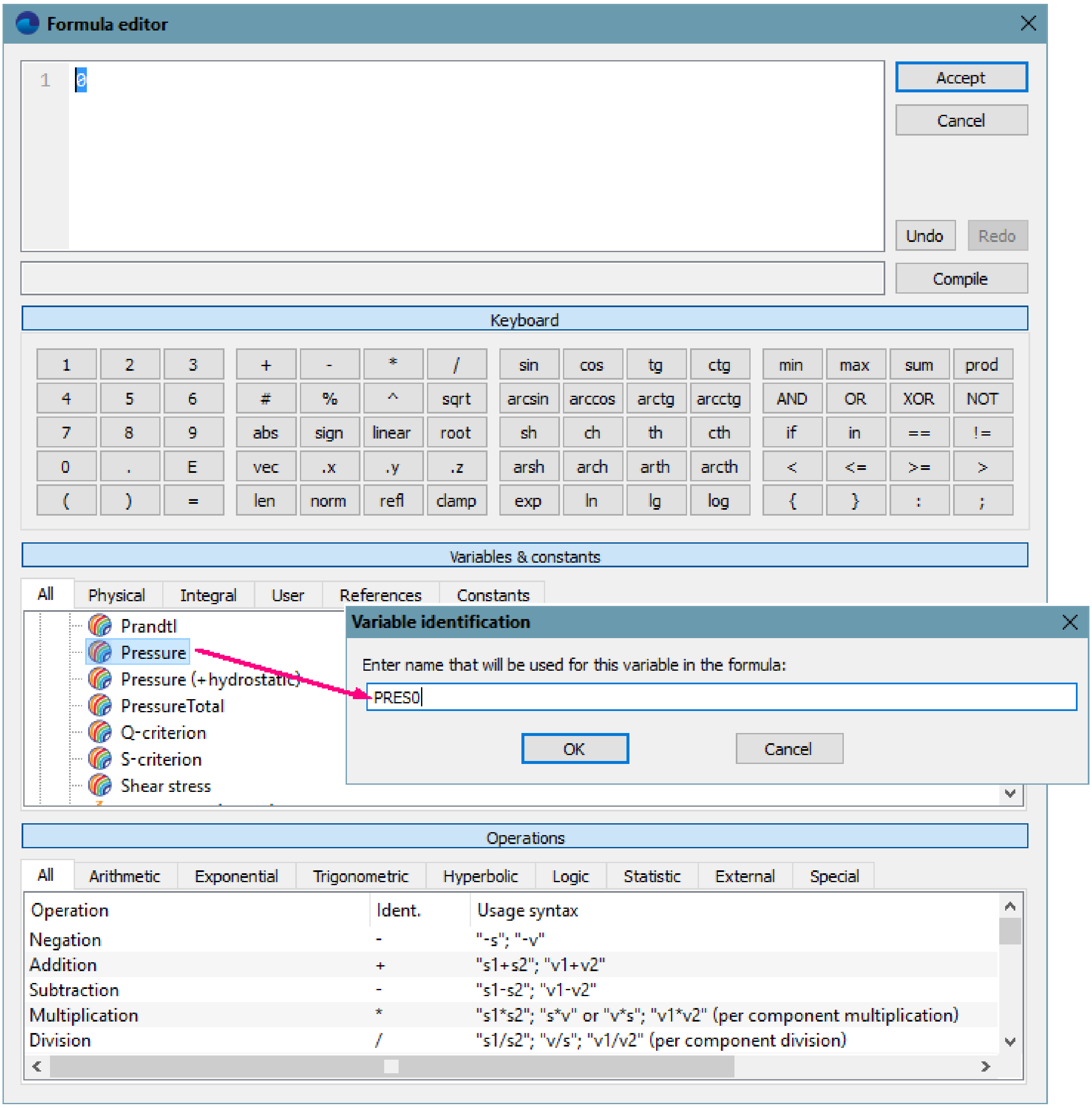
- From the context menu of the folder **User Variables > Local**, select **Create > Scalar**.



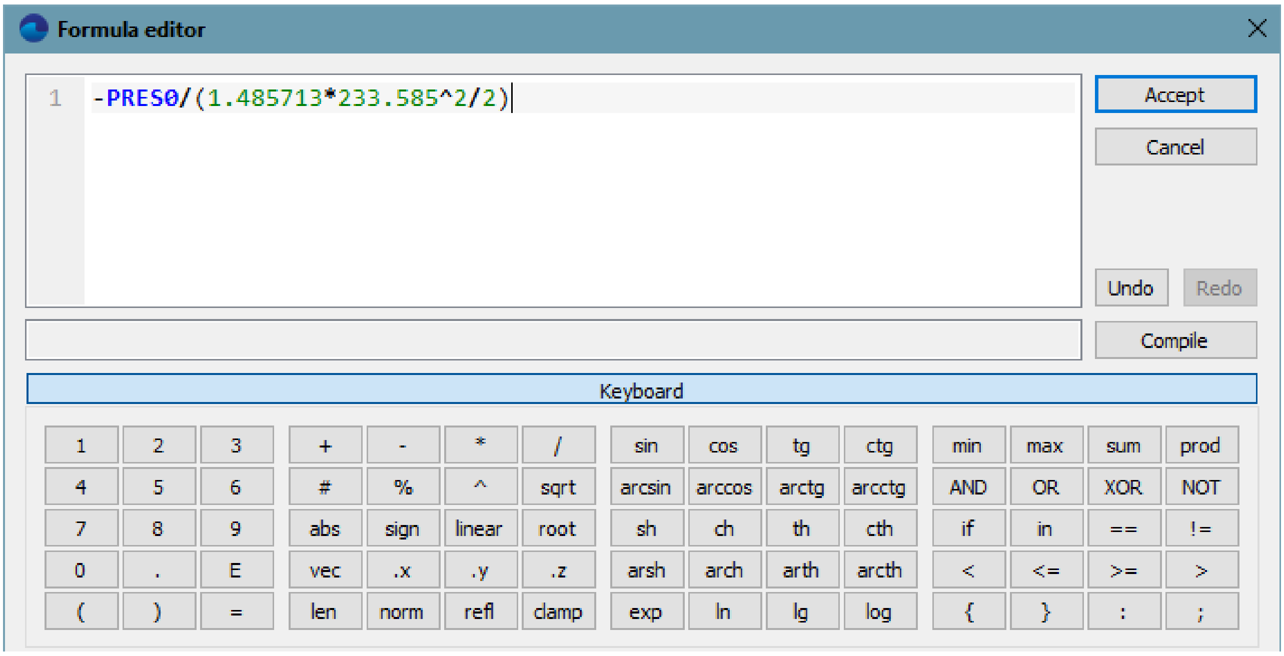
- In the **Properties** window of the just created variable, open the **Formula editor** (select the **Value** field and then click there):



- In the **Formula editor**, identify the variable **Pressure (Phase #0)**.
  - in the pane **Variables & constants**, in the tab **All**, select in **Phase #0** the variable **Pressure** and open the window **Variable identification**.
  - in the **Variable identification** window, specify the name under which the variable will be presented in the formula

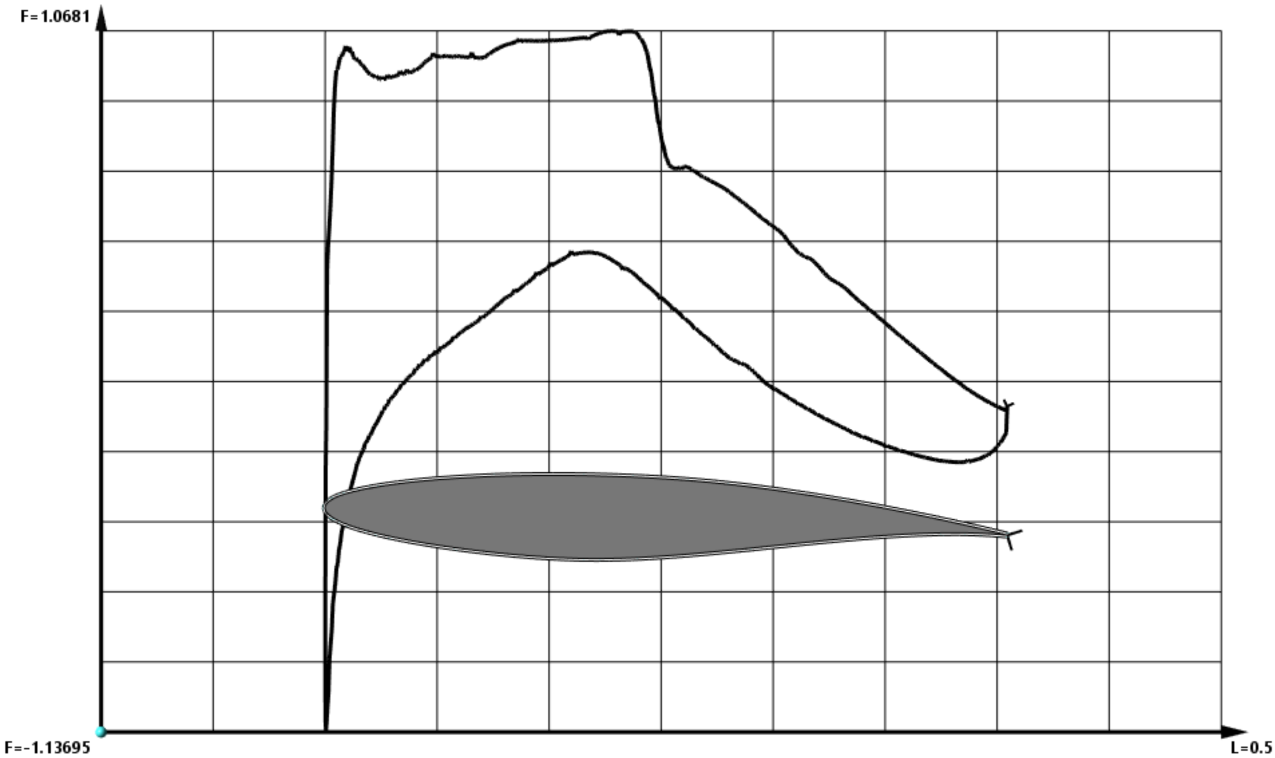


- In the **Formula pane** of the **Formula editor** enter **-PRES0/(1.485713*233.585^2/2)**



- Click **Accept**.

5.3.1.7.2.2 Creating a plot along curve



- In the **Properties** window of **Plane #0**, specify:

Object

Reference point

X	**= -0.1**
Y	**= -0.1**

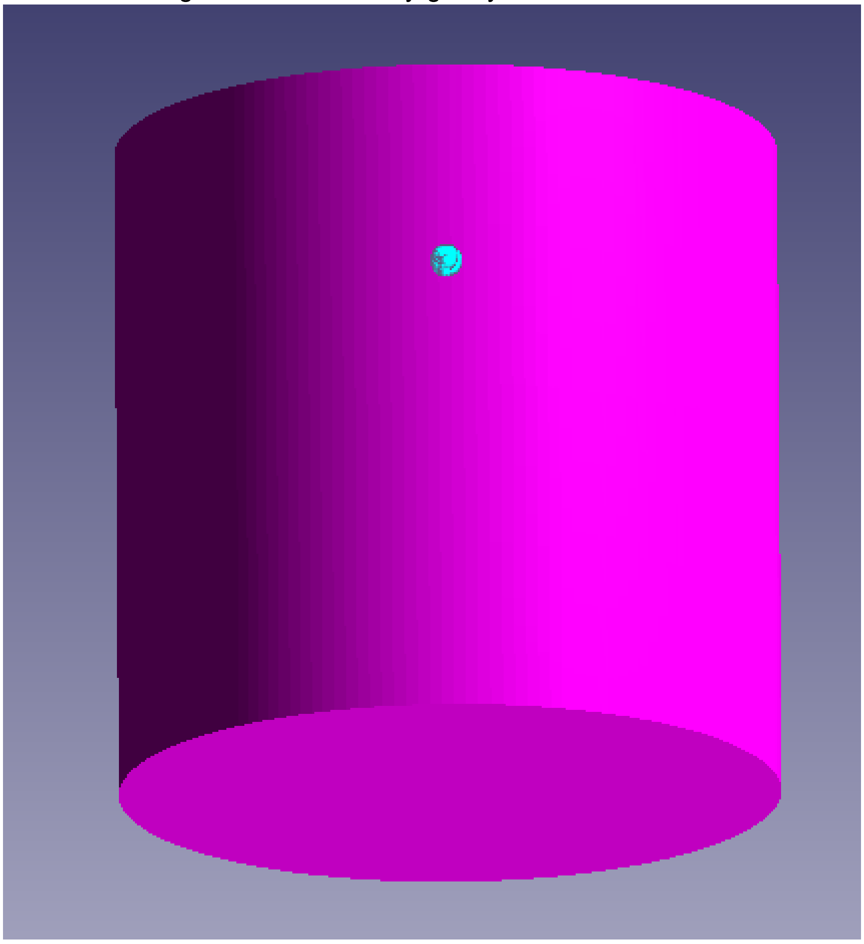
Z = 0.05

- Create a layer **Plot along curve** on **Plane #0**.
- In the **Properties** window of the **Plot along curve**, specify:

Variable	
Category	**= User variables**
Variable	**= UV #0**
Number of points	**= 1000**
Distribute along	**= Every curve**
Rotation angle	**= 90**
Axis X	
Length	
Mode	**= Manual**
Value	**= 0.5**
Appearance	
Plots	
Visible	**= 1**

5.3.2 Ball falling in viscous fluid

In this example, we consider falling of a ball caused by gravity inside a column of viscous incompressible fluid.



Forces act on the ball falling in the fluid, the force of gravity and the force of resistance.

After some time a stationary regime is reached, when the velocity of the ball becomes constant.

Parameters of the problem setting

Dimensions of the region		= 30 × 30	[m × m]
Parameters of the ball			
Radius	R	= 0.5	[m]
Density	ρ	= 1500	[kg m^{-3}]
Fluid parameters:			
Density	ρ	= 1000	[kg m^{-3}]
Viscosity	μ	= 1000	[kg $m^{-1}s^{-1}$]

Geometry **`FallingBall_Domain.STL`**

Project **`Falling_Ball`**

5.3.2.1 Physical model

In the **Properties** window of the element **General settings** specify:

- Add a hydrostatic layer by clicking **Stratum >** .
- Specify the following parameters:

Gravity vector		
X	**= 0**	[m s^{-2}]
Y	**=-9.8**	[m s^{-2}]
Z	**=0**	[m s^{-2}]
g-Point		
X	**= 0**	[m]
Y	**= 5**	[m]
Z	**= 0**	[m]
Stratum		
[0]		
g-Thickness	**= 30**	[m]
g-Density	**= 1000**	[kg m^{-3}]

In the folder **Substances**:

- Create **Substance #0**.
- Specify the following properties of **Substance #0**:

Aggregative state	**= Liquid**	
Molar mass		
Value	**=0.018**	[kg $mole^{-1}$]
Density		
Value	**= 1000**	[kg m^{-3}]
Viscosity		
Value	**= 1000**	[kg $m^{-1}s^{-1}$]
Specific heat		
Value	**= 4217**	[J kg^{-1} K^{-1}]

In the folder **Phases**:

- Create a continuous **Phase #0**.
- In **Phase #0** add **Substance #0** into the folder **Substances**.
- Specify in properties of the folder **Phase #0 > Physical processes**:

Motion **= Navier-Stokes model**

In the folder **Models**:

- Create **Model #0**.
- Add **Phase #0** into subfolder **Model #0 > Phases**.

Note:

The value **Viscosity=1000** is set to increase the convergence.

5.3.2.2 Moving body

In the **Properties** window of **SubRegion #0**, specify:

Model = Model #0.

Create the geometry of the **Moving body**:

- In **Preprocessor**, in the folder **Objects**, create a new **Ellipsoid/sphere** object.
- In the **Properties** window of **Ellipsoid/sphere #0**, specify:

Parameters

Radius	**= 0.5**	[m]

Approximation

Subdivisions	**= 200**

- From the context menu of **Ellipsoid/Sphere #0**, select **Copy as imported object**.

Specify the **Moving body**:

- In the folder **SubRegion #0 > Modifiers** create a **Moving body** based on **Imported object #0**.
- In the **Properties** window of **Moving body #0**, specify:

Mass Properties

Mass [kg]	= 785.4	[kg]

Translation

TimeForces [s]

X	**= 0**	[s]
Y	**= 0**	[s]
Z	**= 0**	[s]

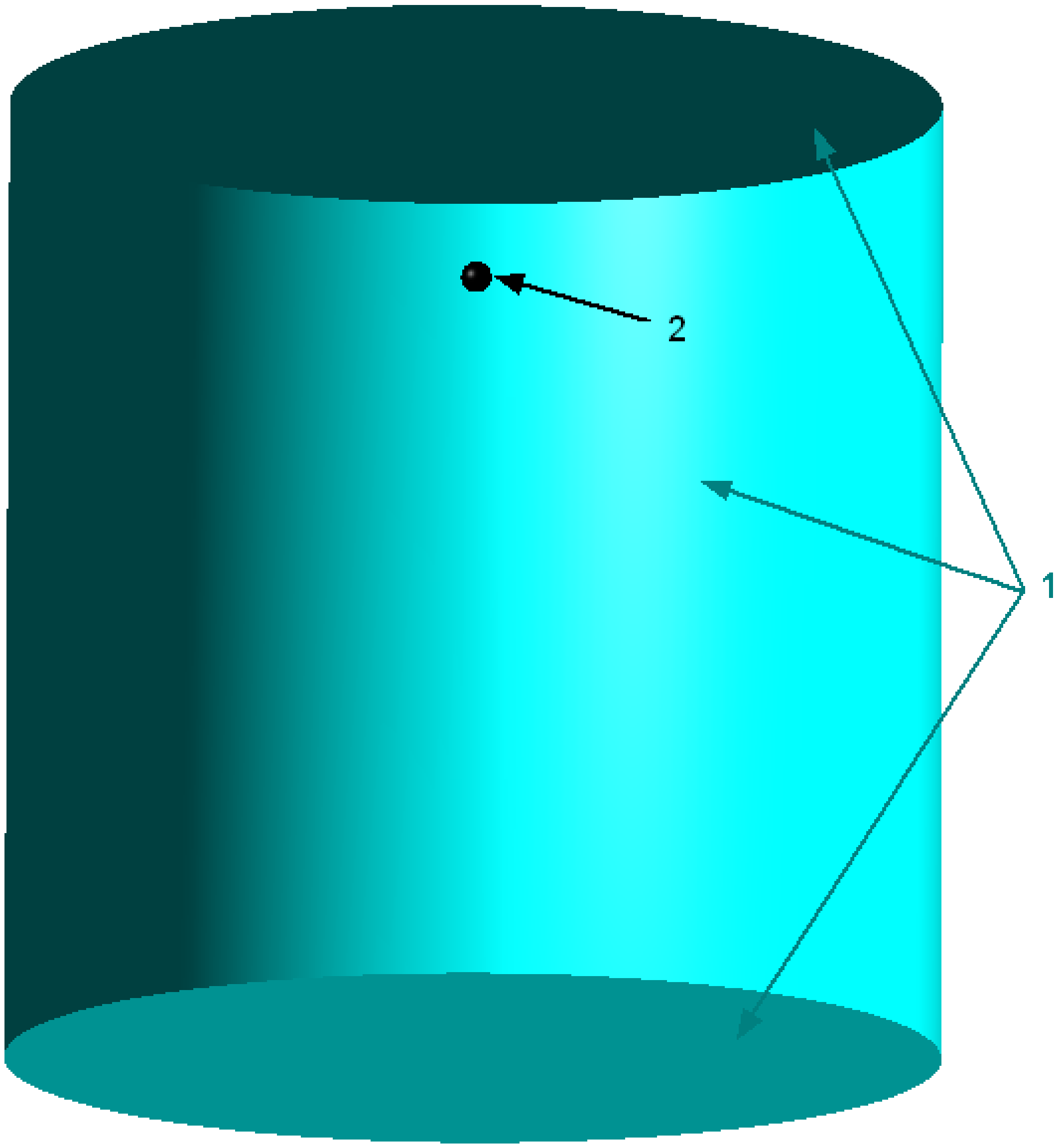
HydroForce [N]

X	**= No**
Y	**= Yes**
Z	**= No**

Notes:

1. Values of **HydroForce** and **HydroTorque** the program calculates automatically.
2. In order to place the body in the initial position, you have to click the button **Operations >**  **(Place to initial position)**.

5.3.2.3 Boundary conditions



Specify the following boundary conditions:

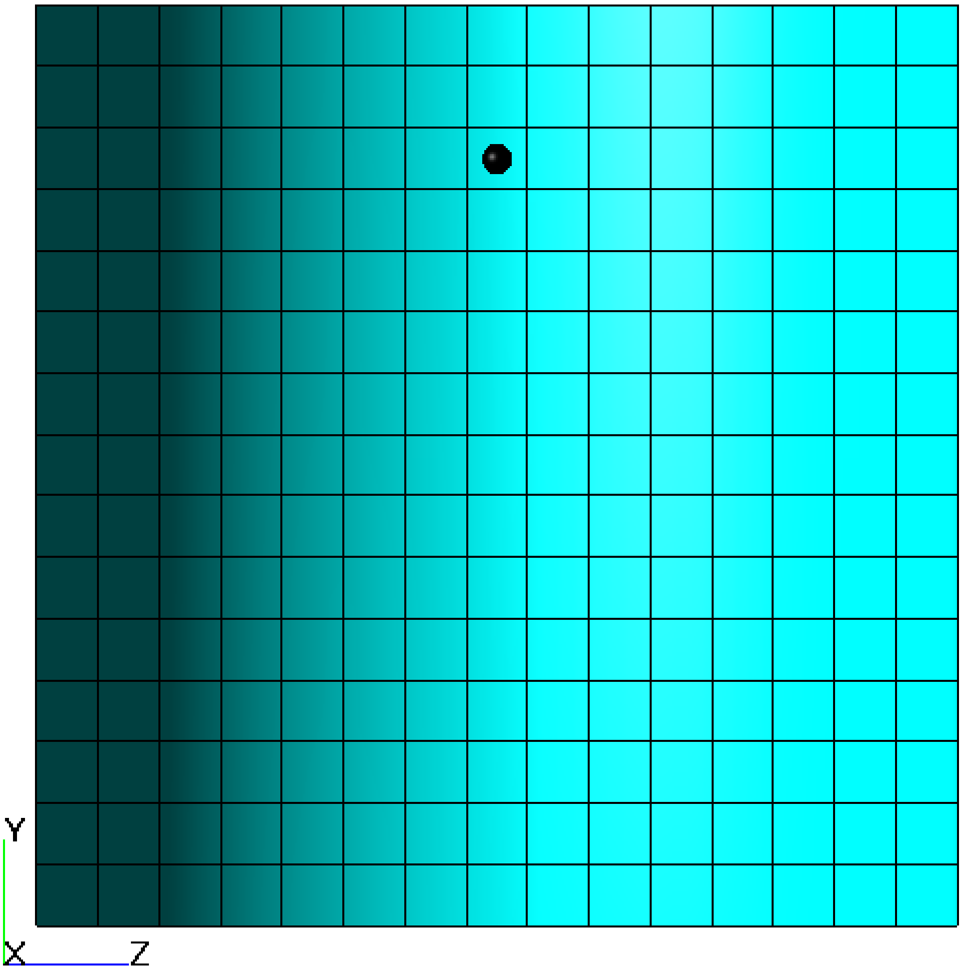
Boundary 1

Type		**= Free Outlet**	
Variables			
	Velocity(Phase #0)	**= Pressure**	
	Value	**= 0**	[Pa]

Boundary 2

Type		**= Wall**
Variables		
	Velocity(Phase #0)	**= No slip**

5.3.2.4 Initial grid



Specify in the **Properties** window of the **Initial grid**:

nX	**= 15**
nY	**= 15**
nZ	**= 15**

In the **Properties** window of the **Initial grid** click **Apply**.

5.3.2.5 Adaptation of the computational grid

In this example, the grid should be adapted on the surface of the moving ball and merge the previously adapted cells that locate away from the ball. To do this, you have to set two **Adaptations**:

1. Splitting the cells on the surface of the ball
2. Merging the cells in the volume away from the ball

Specify the adaptation on the surface of the ball:

- Create **Computational grid > Adaptation > Adaptation #0**.
- Add the boundary condition, which corresponds to the ball's surface, to the subfolder **Computational grid > Adaptation > Adaptation #0 > Objects**.
- In properties of **Adaptation #0** specify:

Enabled	**= Yes**
Max level N	**= 4**
Layers	

Layers for Level N	**= 4**
Layers for Level N-1	**= 4**
Layers for Level N-2	**= 4**
Layers for Level N-3	**= 4**

Specify the adaptation for merge of the previously split cells in the space along from the ball:

- Use the **Copy** command from the context menu of the element **Computational grid > Adaptation > Adaptation #0** to create the element **Computational grid > Adaptation > Adaptation #1**.
- From the context menu of the subfolder **Computational grid > Adaptation > Adaptation #1 > Objects** select the command **Add/Remove Boundary Conditions** and in the **Select boundary conditions** dialog box, which opens, remove the boundary condition, which corresponds to the surface of the ball, from the pane **Selected**, and then click **OK**.
- Add the geometry object **Computational space** to the subfolder **Computational grid > Adaptation > Adaptation #1 > Objects**.
- In properties of **Adaptation #1** specify:

Enabled	**= Yes**
Max level N	**= 0**
Split/Merge	**= Merge**

5.3.2.6 Parameters of calculation

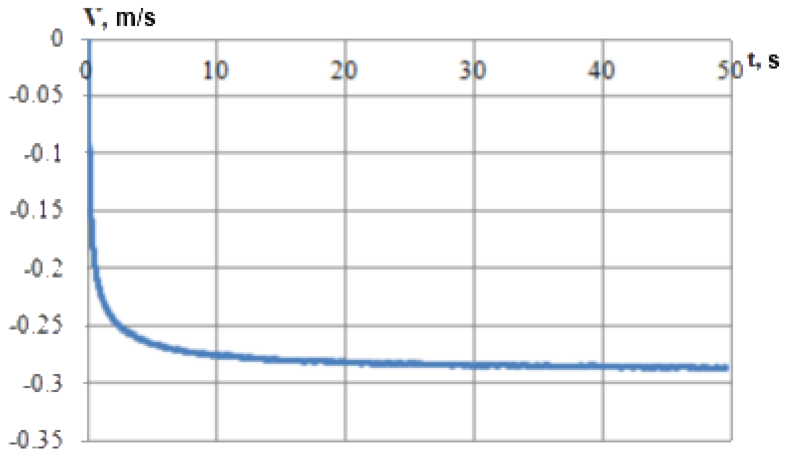
Specify in the **Solver** tab in properties of the **Time step** element:

Method	**= Via CFL number**	
Convective CFL	**= 1**	
Surface CFL	**= 20**	
Max step	**= 0.01**	[s]

5.3.2.7 Visualization

To view the dynamics of the solution during the computation, specify visualization of the **dependency of the ball's velocity on the time** prior the start of computation.

5.3.2.7.1 Ball's velocity in time



- Create **Characteristics** on the **Imported object** in **Postprocessor**.
- In the **Properties** window of the **Characteristics**, specify:

Characteristics

Variable

Variable = **Velocity**

Component = **Y**

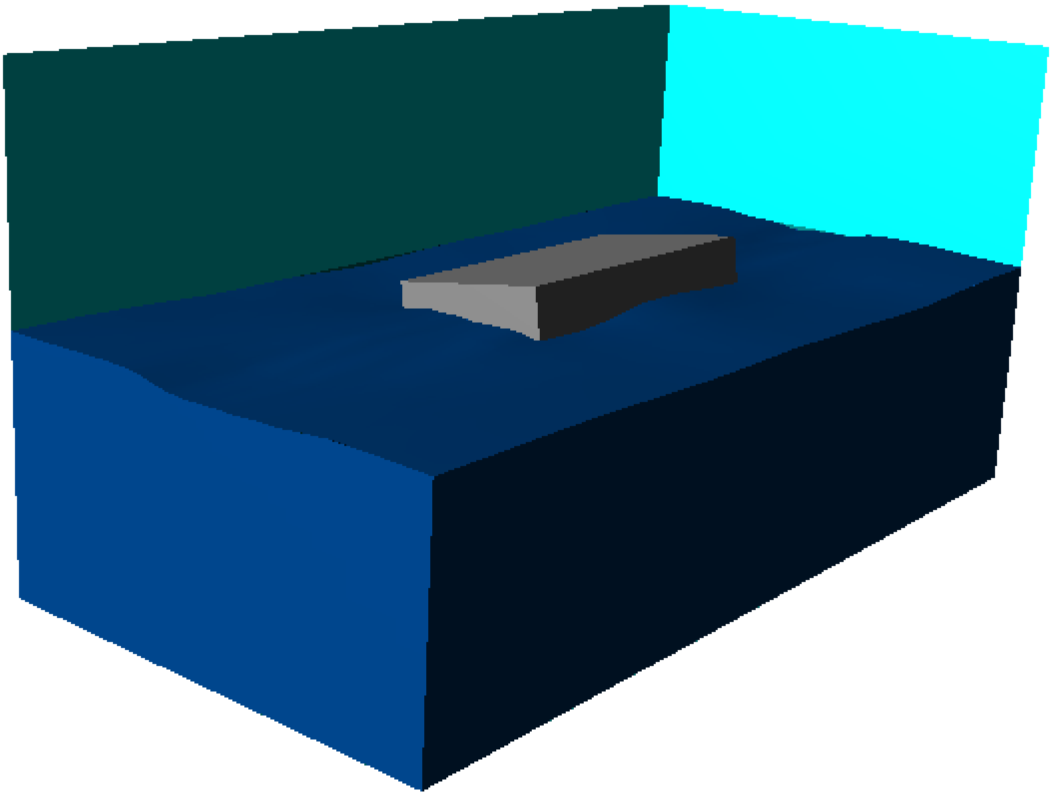
Save to file

Type = **Automatic**

- After the computation is done, download the `glo`-file from the server part of the project using the menu command **File > Download additional files** and selecting the **GLO-files** checkbox.
- Using the data from the `glo`-file, plot the dependency of the ball's velocity (**Velocity.y**) on the time (**Time**).

5.3.3 Floating box

In this example, we consider the motion of a body with a displaced center of mass on the surface of the water and generation of waves on the surface under because of the body's motion.



Parameters of the problem setting

Dimensions of the region		= 0.6 × 0.3 × 0.3	[m × m × m]
Parameters of the body:			
Dimensions	a × b × c	= 0.2 × 0.05 × 0.1	[m × m × m]
Density	ρ	= 500	[kg m^{-3}]
Parameters of the liquid:			
Density	ρ	= 1000	[kg m^{-3}]
Viscosity	μ	= 0.001	[kg m^{-1}s^{-1}]

Geometry	`FloatingBox_Domain.wrl`
Project	`FloatingBox`

5.3.3.1 Physical model

In the **Properties** window of the element **General settings** specify:

- Add a hydrostatic layer by clicking **Stratum >** .
- Specify the following parameters:

Gravity vector			
	X	**= 0**	[m s^{-2}]
	Y	**=-9.8**	[m s^{-2}]
	Z	**=0**	[m s^{-2}]
g-Point			
	X	**= 0**	[m]
	Y	**= 0**	[m]
	Z	**= 0**	[m]
Stratum			
[0]			
	g-Thickness	**= 0.15**	[m]
	g-Density	**= 1000**	[kg m^{-3}]

In the folder **Substances**:

- Create **Substance #0**.
- Specify the following properties of **Substance #0**:

Aggregative state		**= Liquid**	
Molar mass			
	Value	**=0.018**	[kg $mole^{-1}$]
Density			
	Value	**= 1000**	[kg m^{-3}]
	dRho/dP	**= 5.102e-007**	[kg m^{-3} Pa^{-1}]
Viscosity			
	Value	**= 0.001**	[kg $m^{-1}s^{-1}$]
Specific heat			
	Value	**= 4217**	[J kg^{-1} K^{-1}]

In the folder **Phases**:

- Create a continuous **Phase #0**.
- In **Phase #0** add **Substance #0** into the folder **Substances**.
- Specify in properties of the folder **Phase #0 > Physical processes**:

Motion	**= Navier-Stokes model**

- Create a continuous **Phase #1**.

In the folder **Models**:

- Create **Model #0**.
- Add **Phase #0** and **Phase #1** into subfolder **Model #0 > Phases**.

In properties of **SubRegion #0** specify:

Model = Model #0

5.3.3.2 Moving body

Create a **Moving body**:

- Download the geometry of an **Imported object** from the file `FloatingBox_Box.stl`.
- In the folder **SubRegion #0 > Modifiers** create a **Moving body** on **Imported object #0**.

In the **Properties** window of the **Moving body**, specify:

Mass Properties		
Mass [kg]	**= 0.5**	[kg]
Center of Inertia		
X	**= 0**	[m]
Y	**= 0**	[m]
Z	**= -0.005**	[m]
Moment Inertia X [kg*m2]		
X	**= 0.00846**	[kg m^2]
Y	**= 0**	[kg m^2]
Z	**= 0**	[kg m^2]
Moment Inertia Y [kg*m2]		
X	**= 0**	[kg m^2]
Y	**= 0.00333**	[kg m^2]
Z	**= 0**	[kg m^2]
Moment Inertia Z [kg*m2]		
X	**= 0**	[kg m^2]
Y	**= 0**	[kg m^2]
Z	**= 0.00333**	[kg m^2]
Translation		
TimeForces [s]		
X	**= 0**	[s]
Y	**= 0**	[s]
Z	**= 0**	[s]
HydroForce [N]		
X	**= No**	
Y	**= Yes**	
Z	**= No**	
Rotation		
TimeTorques [s]		
X	**= 0**	[s]
Y	**= 0**	[s]
Z	**= 0**	[s]
HydroTorque [N*m]		
X	**= Yes**	
Y	**= No**	
Z	**= No**	

Initial position

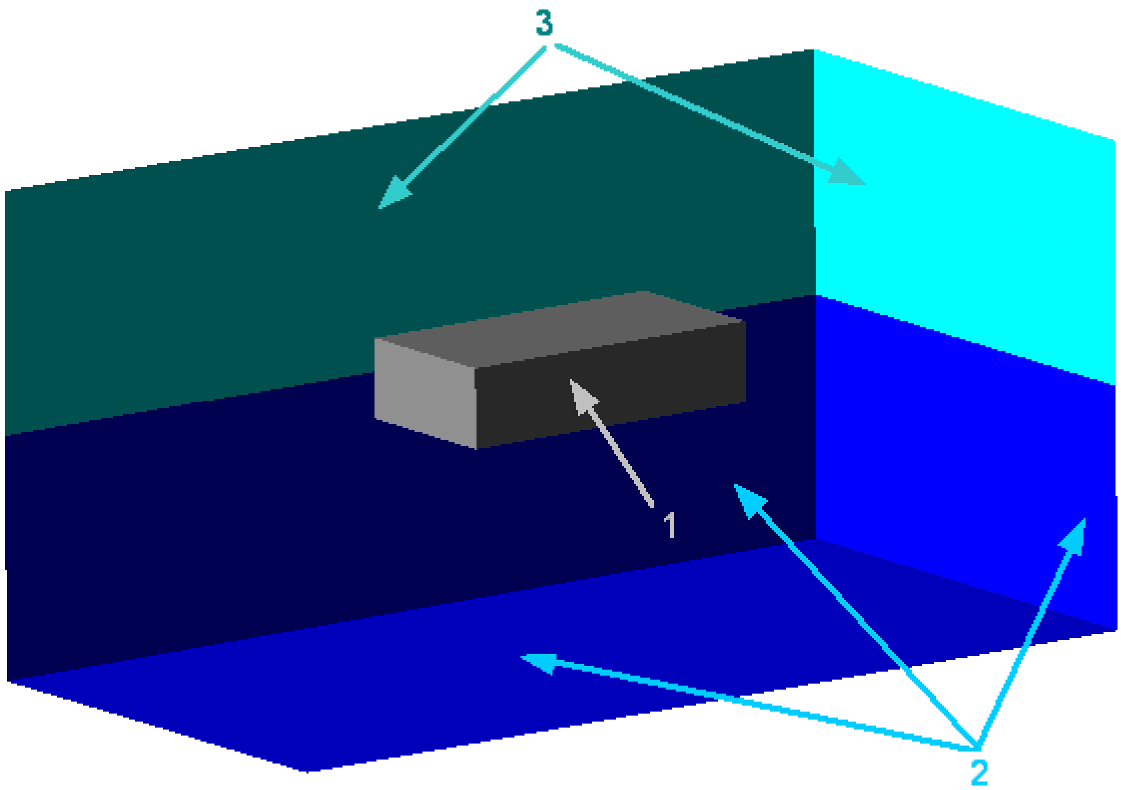
Reference point

X	**= 0.3**	[m]
Y	**= 0.05**	[m]
Z	**= 0.15**	[m]

Note:

In order to place the body in the initial position, you have to click the button **Operations >** (**Place to initial position**).

5.3.3.3 Boundary conditions



Specify the following boundary conditions:

Boundary 1

Type	**= Wall**
Variables	
Velocity(Phase #0)	**= No slip**
VOF(Phase #0)	**= Symmetry**

Boundary 2

Type	**= Free Outlet**
Variables	
Velocity (Phase #0)	**= Pressure**
Value	**= 0**
VOF (Phase #0)	**= Value**
Value	**= 1**

Boundary 3

Type = **Free Outlet**

Variables

Velocity (Phase #0) = **Pressure**

Value = **0**

VOF (Phase #0) = **Value**

Value = **0**

5.3.3.4 Initial conditions

In **Model #0**, in **Init. data #0**, specify:

VOF

Value = **1**

In the folder **Objects**:

- create **Box #0**.
- In the **Properties** window of **Box #0** specify:

Location

Reference point

X	**= 0.3**	[m]
Y	**= -0.075**	[m]
Z	**= 0.15**	[m]

Size

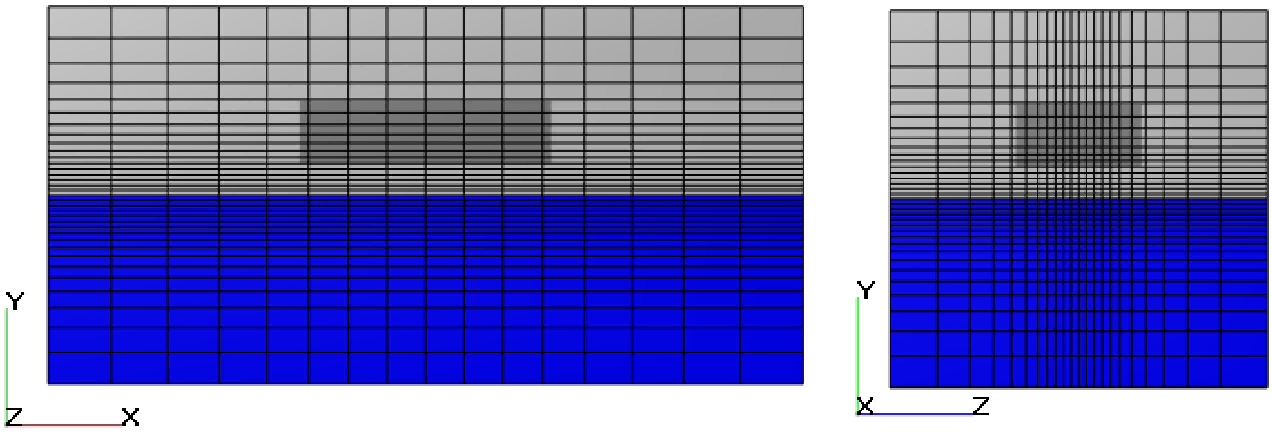
X	**= 0.6**	[m]
Y	**= 0.149**	[m]
Z	**= 0.3**	[m]

In **SubRegion #0**, in the **Properties** window of the element **Initial conditions > Init. condition #0**, specify:

Object = **Box #0**

Init. data = **Init. data #0**

5.3.3.5 Initial grid



In the **Properties** window of the **Initial grid**, click the button to open the **Initial grid editor**.

Specify in the **Initial grid editor**:

for axis OX

Grid parameters

h_max **= 0.05** [m]

h_min **= 0.03** [m]

Insert a reference line with a coordinate **x=0.3** [m].

Specify **Reference line parameters** for the reference line with coordinate **x=0**:

h **= 0.05** [m]

Specify **Reference line parameters** for the reference line with coordinate **x=0.3**:

h **= 0.03** [m]

Specify **Reference line parameters** for the reference line with coordinate **x=0.6**:

h **= 0.05** [m]

for axis OY (click the button Y)

Grid parameters

h_max **= 0.025** [m]

h_min **= 0.004** [m]

Insert a reference line with a coordinate **y=0** [m].

Specify **Reference line parameters** for the reference line with coordinate **y=-0.15**:

h **= 0.025** [m]

Specify **Reference line parameters** for the reference line with coordinate **y=0**:

h **= 0.004** [m]

Specify **Reference line parameters** for the reference line with coordinate **y=0.15**:

h **= 0.025** [m]

for axis OZ (click the button Z)

Grid parameters

h_max **= 0.0375** [m]

h_min **= 0.006** [m]

Insert a reference line with a coordinate **z=0.15** [m].

Specify **Reference line parameters** for the reference line with coordinate **z=0**:

h **= 0.0375** [m]

Specify **Reference line parameters** for the reference line with coordinate **z=0.15**:

h **= 0.006** [m]

Specify **Reference line parameters** for the reference line with coordinate **z=0.3**:

h **= 0.0375** [m]

Click **OK** to close the **Initial grid editor** with saving the entered data.

In the **Properties** window of the **Initial grid** click **Apply**.

5.3.3.6 Parameters of calculation

Specify in the **Solver** tab in properties of the **Time step** element:

Method **= Via CFL number**

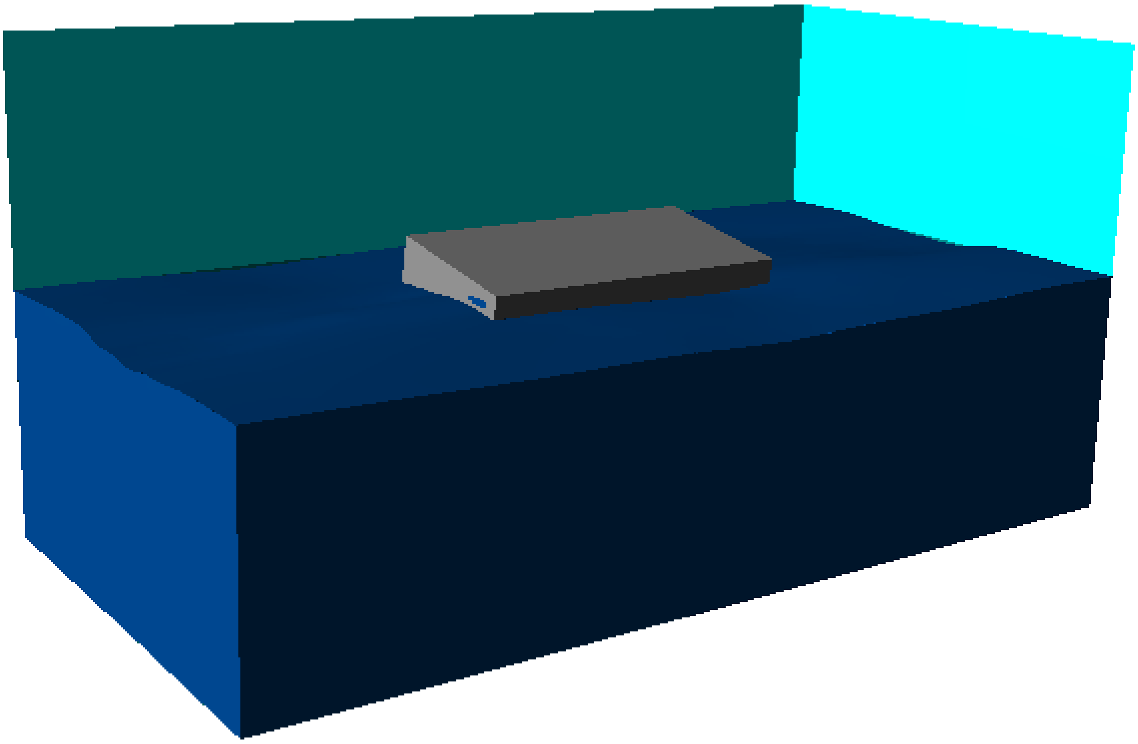
Surface CFL **= 1**

Max step **= 0.1** [s]

5.3.3.7 Visualization

To view the dynamics of the solution during the computation, prior to computation, specify visualization of a **Layer** for water surface.

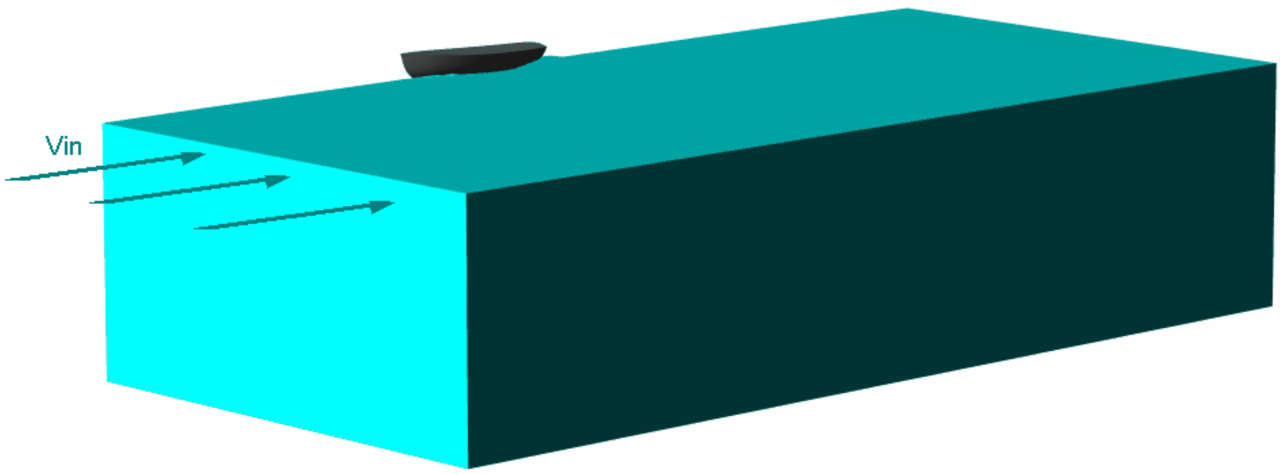
5.3.3.7.1 Water surface



- Create a **VOF** layer on **Computational space**.

5.3.4 Floating boat

In this example, we simulate motion of a boat through the water.



Parameters of the problem setting

Dimensions of the region	= 52 × 25 × 24	[m × m × m]

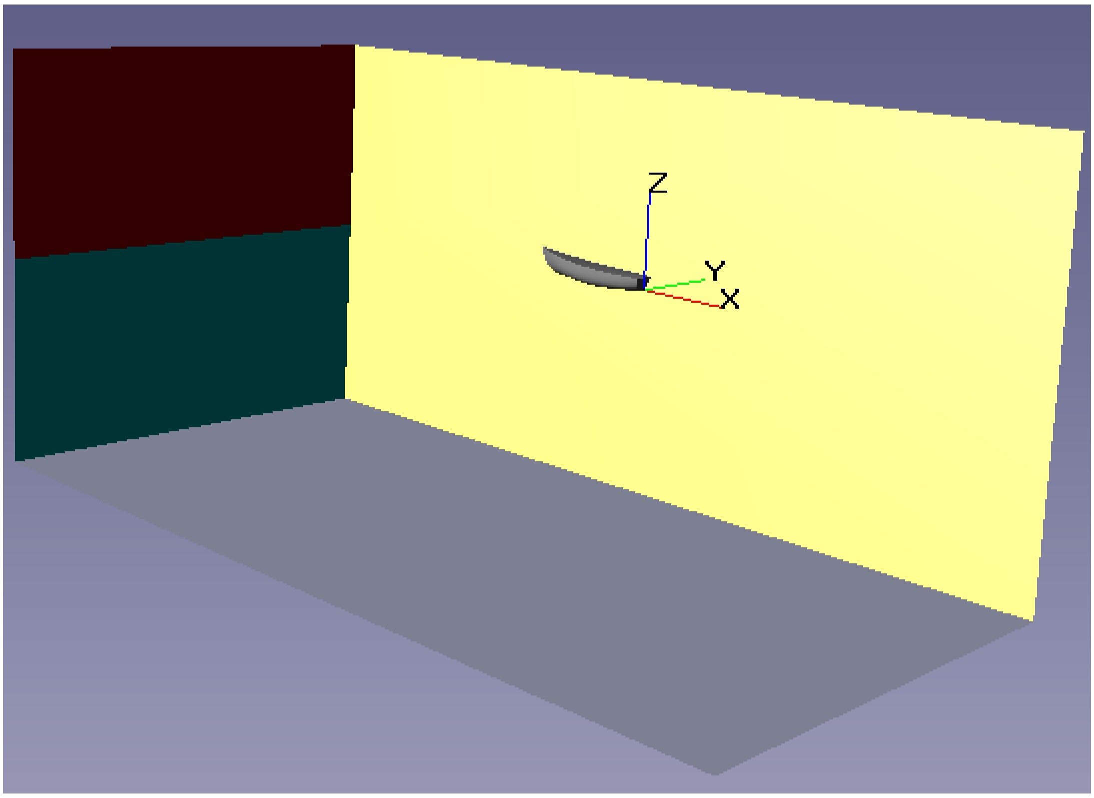
Parameters of the body:

Mass	m	= 968	[kg]
Parameters of water:			
Density	ρ	= 1000	[kg m^{-3}]
Viscosity	μ	= 0.001	[kg $m^{-1}s^{-1}$]
Inlet rate	V_{in}	= 4	[m s^{-1}]
Geometry		`Boat_Domain.wrl`	
Project		`Boat`	

Note:

Calculation of the project may require significant computing resources and long time.

5.3.4.1 Physical model



In the **Properties** window of the element **General settings** specify:

- Add a hydrostatic layer by clicking **Stratum >** .
- Specify the following parameters:

Gravity vector			
X	**= 0**	[m s^{-2}]	
Y	**=0**	[m s^{-2}]	
Z	**=-9.8**	[m s^{-2}]	
g-Point			
X	**= 0**	[m]	
Y	**= 0**	[m]	
Z	**= 0**	[m]	
Stratum			
[0]			

g-Thickness	**= 12**	[m]
g-Density	**= 1000**	[kg m^{-3}]

In the folder **Substances**:

- Create **Substance #0**.
- Specify the following properties of **Substance #0**:

Aggregative state		**= Liquid**	
Molar mass			
	Value	**=0.018**	[kg $mole^{-1}$]
Density			
	Value	**= 1000**	[kg m^{-3}]
Viscosity			
	Value	**= 0.001**	[kg $m^{-1}s^{-1}$]
Specific heat			
	Value	**= 4217**	[J kg^{-1} K^{-1}]

In the folder **Phases**:

- Create a continuous **Phase #0**.
- In **Phase #0** add **Substance #0** into the folder **Substances**.
- Specify in properties of the folder **Phase #0 > Physical processes**:

Motion	**= Navier-Stokes model**
Turbulence	**= KES**

- Create a continuous **Phase #1**.

In the folder **Models**:

- Create **Model #0**.
- Add **Phase #0** and **Phase #1** into subfolder **Model #0 > Phases**.

In properties of **SubRegion #0** specify:

Model = Model #0

5.3.4.2 Moving body

Create a **Moving body**:

- Download the geometry of an **Imported object** from the file `Boat_Body.wrl`.
- In the folder **SubRegion #0 > Modifiers** create a **Moving body** on **Imported object #0**.

In the **Properties** window of the **Moving body**, specify:

Mass Properties

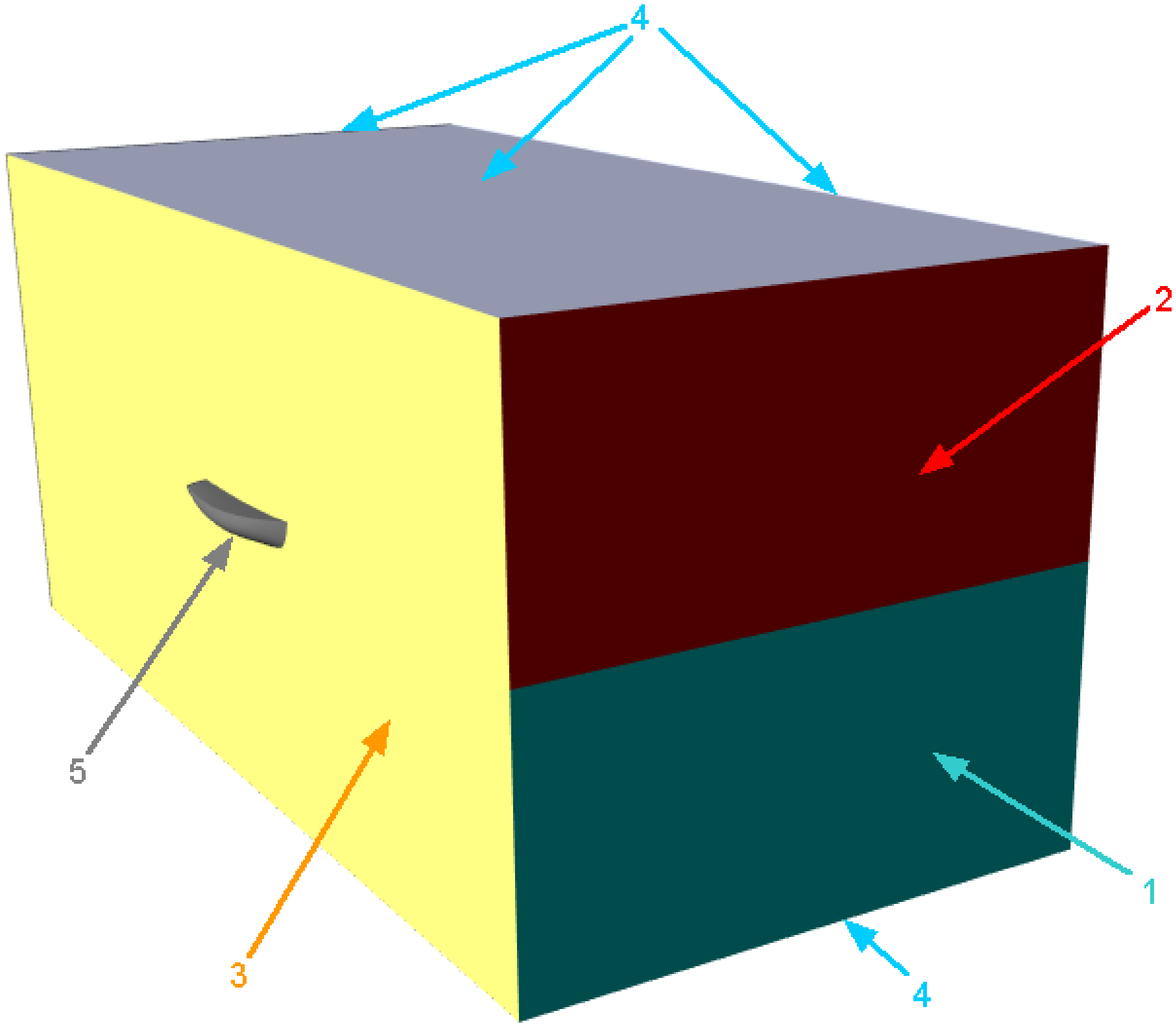
Mass [kg]		**= 968**	[kg]
Center of Inertia			
	X	**= 3.856**	[m]
	Y	**= 0**	[m]
	Z	**= -0.12**	[m]
Moment Inertia X [kg*m2]			

X	= 304	[kg m^2]
Y	= 0	[kg m^2]
Z	= 0	[kg m^2]
Moment Inertia Y [kg*m2]		
X	= 0	[kg m^2]
Y	= 6025	[kg m^2]
Z	= 0	[kg m^2]
Moment Inertia Z [kg*m2]		
X	= 0	[kg m^2]
Y	= 0	[kg m^2]
Z	= 6080	[kg m^2]
Translation		
TimeForces [s]		
X	= 0	[s]
Y	= 0	[s]
Z	= 0	[s]
HydroForce [N]		
X	= No	
Y	= No	
Z	= Yes	
Rotation		
TimeTorques [s]		
X	= 0	[s]
Y	= 0	[s]
Z	= 0	[s]
HydroTorque [N*m]		
X	= No	
Y	= Yes	
Z	= No	
Initial position		
Reference point		
X	= 0	[m]
Y	= 0	[m]
Z	= 0	[m]
Rotation definition	By matrix	
Axis X		
X	= -1	
Y	= 0	
Z	= 0	
Axis Y		
X	= 0	
Y	= -1	
Z	= 0	
FSI		
Artificial compressibility	= Yes	

Flexibility	**= 0.0001**	[m Pa^{-1}]
Mobility	**= 0.05**	[m^2kg^{-1}]

- In the **Properties** window of the **Moving body**, click **Operations >**  **(Place to initial position)**.

5.3.4.3 Boundary conditions



Specify the following boundary conditions:

Boundary 1

Type	**= Inlet/Outlet**	
Variables		
Velocity(Phase #0)	**= Normal mass velocity**	
Mass velocity	**= 4000**	[kg m^{-2} s^{-1}]
TurbEnergy(Phase #0)	**= Pulsations**	
Value	**= 0.01**	
TurbDissipation(Phase #0)	**= Turbulent scale**	
Value	**= 0.01**	[m]
VOF (Phase #0)	**= Value**	
Value	**= 1**	

Boundary 2

Type	**= Inlet/Outlet**

Variables

Velocity(Phase #0)	**= Normal mass velocity**	[kg $m^{-2}s^{-1}$]
Mass velocity	**= 4000**	
TurbEnergy(Phase #0)	**= Pulsations**	
Value	**= 0.01**	
TurbDissipation(Phase #0)	**= Turbulent scale**	
Value	**= 0.01**	[m]
VOF (Phase #0)	**= Zero gradient**	

Boundary 3

Type	**= Symmetry**
Variables	
Velocity(Phase #0)	**= Slip**
TurbEnergy(Phase #0)	**= Symmetry**
TurbDissipation(Phase #0)	**= Symmetry**
VOF (Phase #0)	**= Symmetry**

Boundary 4

Type	**= Free Outlet**	
Variables		
Velocity(Phase #0)	**= Pressure**	
Value	**= 0**	[Pa]
TurbEnergy(Phase #0)	**= Pulsations**	
Value	**= 0.01**	
TurbDissipation(Phase #0)	**= Turbulent scale**	
Value	**= 0.01**	[m]
VOF (Phase #0)	**=Zero gradient**	

Boundary 5

Type	**= Wall**
Variables	
Velocity(Phase #0)	**= Logarithm law**
TurbEnergy(Phase #0)	**= Value in cell near wall**
TurbDissipation(Phase #0)	**= Value in cell near wall**
VOF(Phase #0)	**= Symmetry**

5.3.4.4 Initial conditions

In **Model #0**, in **Init. data #0**, specify:

Velocity(Phase #0)

Value

X	**= 4**	[m s^{-1}]
Y	**= 0**	[m s^{-1}]
Z	**= 0**	[m s^{-1}]

Pulsations(Phase #0)

Value **= 0.01**

Turbulent scale(Phase #0)

Value	**= 0.01**	[m]

VOF(Phase #0)

Value	**= 1**

In the folder **Objects**:

- create **Box #0**.
- In the **Properties** window of **Box #0** specify:

Location

Reference point

X	**= 0**	[m]
Y	**= -12.5**	[m]
Z	**= -6**	[m]

Size

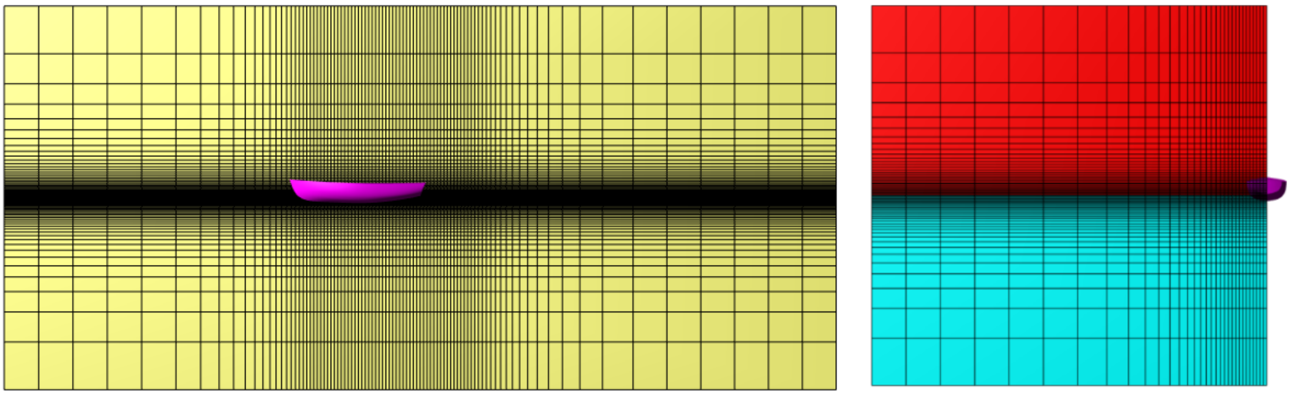
X	**= 52**	[m]
Y	**= 24.999**	[m]
Z	**= 12**	[m]

In **SubRegion #0**:

- In the **Properties** window of the element **Initial conditions > Init. condition #0**, specify:

Object	**= Box #0**
Init. data	**= Init. data #0**

5.3.4.5 Initial grid



For correct simulation of this task you need to make a resolution by the computational grid the free surface and area near the board of the boat.

You do not have to make resolution of areas over the boat and located away from the boat. The outer boundaries of the computational domain locate far away from the boat only to reduce influence of boundary conditions on the solution near the boat.

In this situation it is desirable to condense the initial grind near the free surface (along the axis OZ). Along the axis OY you should condense the grid near the surface of the boat and smoothly increase the size of the cells when moving away from the boat at the distances more then 1.5 of the boat's width. Along the axis OX it is desirable to resolute the boat's head and the aft, where the intensive generation of waves occurs, while along the boat, between its head and the aft, the cells can be slightly elongated because contours of the ship there are uniform and velocity vectors are directed along these elongated cells, so the elongation of the cells does not affect substantially on the simulation's results.

In the **Properties** window of the **Initial grid**, click the button to open the **Initial grid editor**.

Specify in the **Initial grid editor**:

for axis OX

Grid parameters

h_max	**=2**	[m]
h_min	**= 0.2**	[m]

Insert a reference line with coordinate **x=0** [m].

Specify **Reference line parameters** for the reference line with coordinate **x=-26**:

h	**= 2**	[m]
kh+	**= 1.3**	

Specify **Reference line parameters** for the reference line with coordinate **x=0**:

h	**= 0.2**	[m]
kh-	**= 1.3**	
kh+	**= 0.92**	

Specify **Reference line parameters** for the reference line with coordinate **x=26**:

h	**= 2**	[m]
kh-	**= 0.7**	

for axis OY (click the button Y)

Grid parameters

h_max	**= 2**	[m]
h_min	**= 0.2**	[m]

Specify **Reference line parameters** for the reference line with coordinate **y=-25**:

h	**= 2**	[m]
kh+	**= 1.4**	

Specify **Reference line parameters** for the reference line with coordinate **y=0**:

h	**= 0.2**	[m]
kh-	**= 1.06**	

for axis OZ (click the button Z)

Grid parameters

h_max	**=2**	[m]
h_min	**= 0.2**	[m]

Insert a reference line with coordinate **z=0** [m].

Specify **Reference line parameters** for the reference line with coordinate **z=-12**:

h	**= 2**	[m]
kh+	**= 1.05**	

Specify **Reference line parameters** for the reference line with coordinate **z=0**:

h	**= 0.2**	[m]
kh-	**= 1.15**	
kh+	**= 0.8**	

Specify **Reference line parameters** for the reference line with coordinate **z=12**:

h	**= 2**	[m]
kh-	**= 0.95**	

Click **OK** to close the **Initial grid editor** with saving the entered data.

In the **Properties** window of the **Initial grid** click **Apply**.

5.3.4.6 Adaptation of the computational grid

Specify the adaptation on the surface of the boat:

- Create **Computational grid > Adaptation > Adaptation #0**.
- Add the boundary condition, which corresponds to the boat's surface, to the subfolder **Computational grid > Adaptation > Adaptation #0 > Objects**.
- In properties of **Adaptation #0** specify:

Enabled	**= Yes**
Max level N	**= 1**
Layers	
Layers for Level N	**= 4**

Specify the merge of the previously split cells:

- Create **Computational grid > Adaptation > Adaptation #1**.
- Add the geometry object **Computational space** to the subfolder **Computational grid > Adaptation > Adaptation #1 > Objects**.
- In properties of **Adaptation #1** specify:

Enabled	**= Yes**
Max level N	**= 0**
Split/Merge	**= Merge**

5.3.4.7 Parameters of calculation

Specify in the **Solver** tab in properties of the **Time step** element:

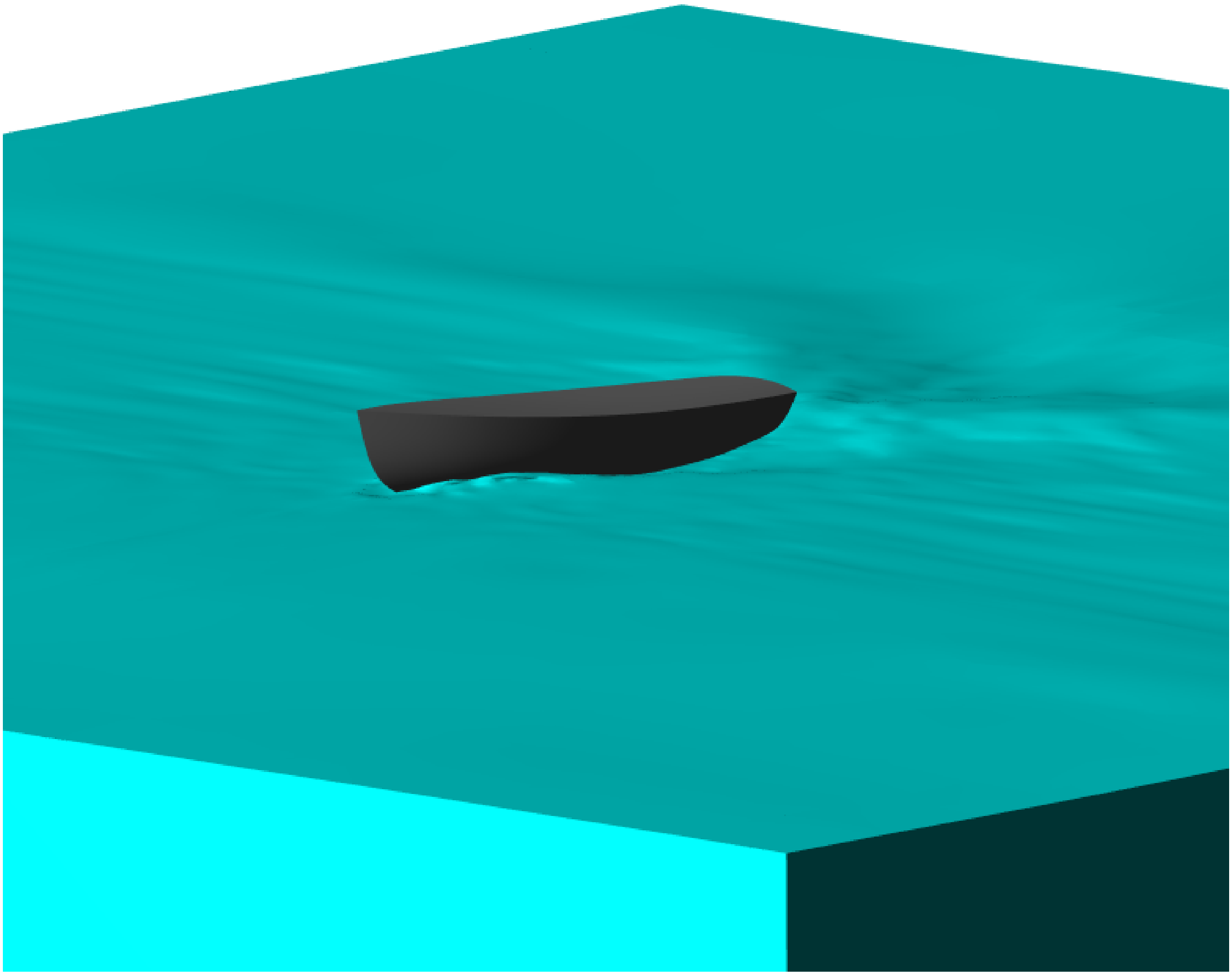
Method	**= Via CFL number**	
Surface CFL	**= 1**	
Max step	**= 1**	[s]

5.3.4.8 Visualization

To view the dynamics of the solution during the computation, prior to computation, specify visualization of:

1. **Water surface**
2. **Pressure distribution** on the surface of the boat

5.3.4.8.1 Water surface



- Create a **VOF** layer on **Computational space**.
- In the **Properties** window of the **VOF** layer, specify:

Appearance			
	Fill		
		Color	**= Aqua**

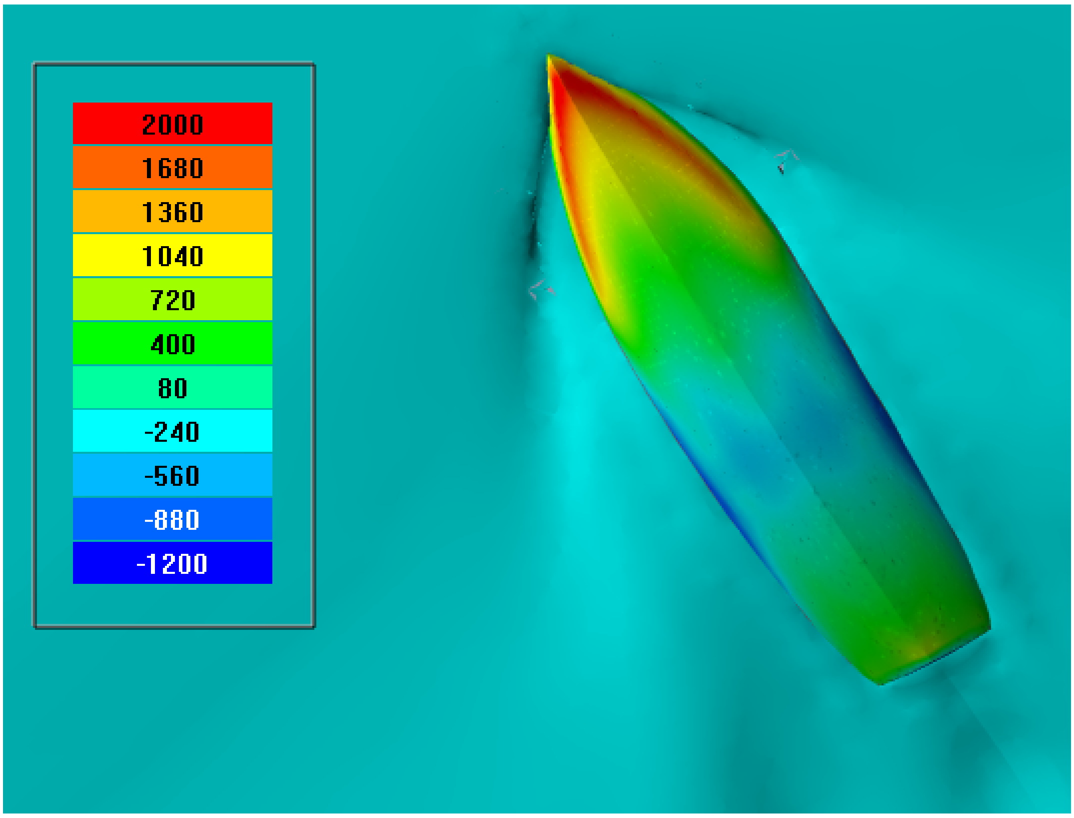
- In the **Properties** window of **Plane #0**, specify:

Object				
	Reference point			
		X	**= 0**	[m]
		Y	**= -0.0001**	[m]
		Z	**= 0**	[m]
	Normal			
		X	**= 0**	
		Y	**= -1**	
		Z	**= 0**	
Clipping object			**= Yes**	
Mirror			**= Yes**	

- In the **Properties** window of the layer **Solids**, specify:

Clipped = **Yes**

5.3.4.8.2 Pressure distribution



- Create a layer **Color contours** on the **Imported object**.
- In the **Properties** window of these **Color contours**, specify:

Variable
- **Variable** = **Pressure**

Shift = **0.000001**

Value range
- **Mode** = **Manual**
- **Max** = **2000**
- **Min** = **-1200**

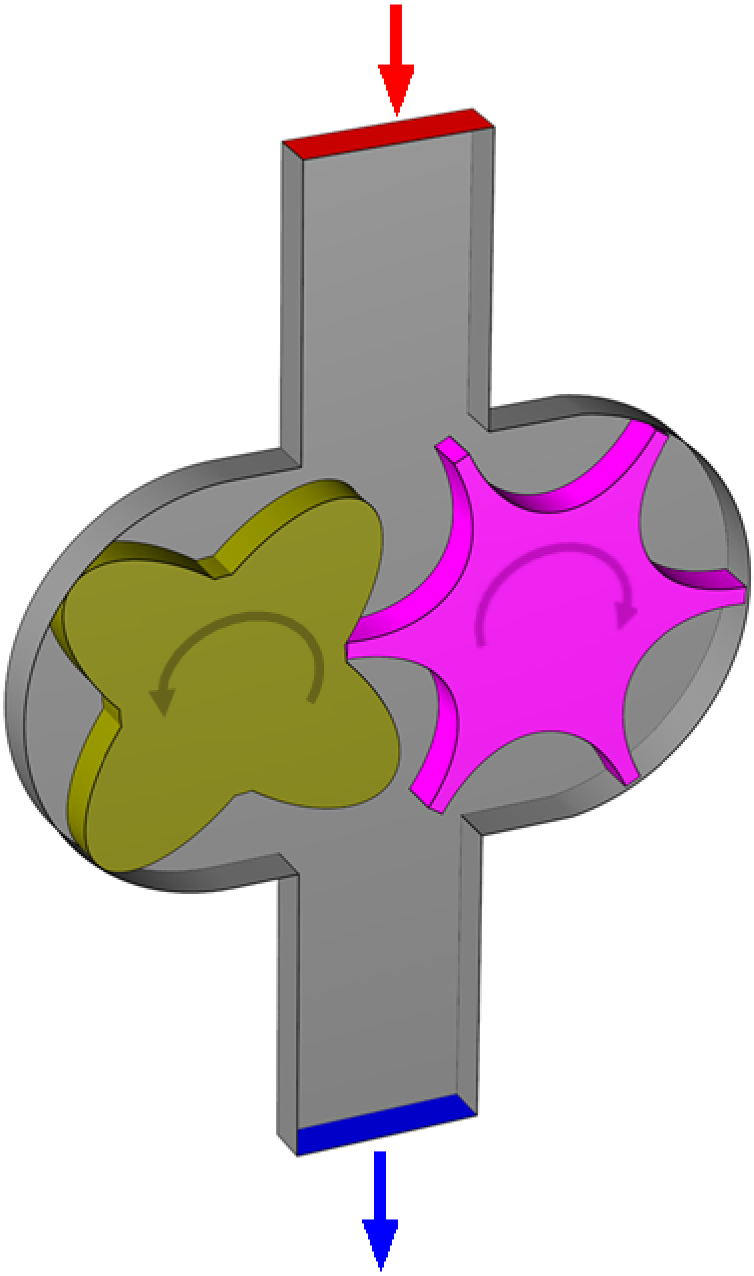
Palette
- **Appearance**
  - **Enabled** = **Yes**
  - **Style** = **Style 1**

Note:

In order to display only the surface of the interface and hide the surface between the phase and the boundary of the computational domain, it is necessary to specify **Volume=No** in properties of the layer **VOF**.

5.3.5 Rotary compressor

In this example, we simulate the flow in a hydraulic pump.



Parameters of the problem setting

Inflow parameters:

Pressure on inlet	p_{in}	= 101000	[Pa]
Back pressure on outlet	p_{out}	= 102000	[Pa]

Parameters of rotors:

Angular velocity of the left rotor	W_l	= 600	[radian s^{-1}]
Angular velocity of the right rotor	W_r	= 400	[radian s^{-1}]

Substance = Air

Geometry `Compressor_Domain.STL`

Project `Compressor`

5.3.5.1 Physical model

In the folder **Substances**:

- Create **Substance #0**.
- Load **Substance #0** from the **Standard** substance database:
  - From the context menu of **Substance #0** select **Load from SD > Standard**.
  - In the new window **Load from database**, select:

Substances **= Air**
Phases **= Gas (equilibrium)**

In the folder **Phases**:

- Create a continuous **Phase #0**.
- Add the **Air_Gas (equilibrium)** substance into the folder **Phase #0 > Substances**.
- Specify in properties of the folder **Physical processes**:

Heat transfer **= Heat transfer via h**
Motion **= Navier-Stokes model**
Turbulence **= KES**

In the folder **Models**:

- Create **Model #0**.
- Add **Phase #0** into subfolder **Model #0 > Phases**.
- In the **Properties** window of **Model #0**, specify:

Use Gap model **= Standard Gap model**

In properties of **SubRegion #0** specify:

Model = Model #0

5.3.5.2 Moving bodies

Create the **Left rotor**:

- Select **Create** from the context menu of the **Objects** folder and create an **Imported object** by loading its geometry from the file `Compressor_Male.wrl`.
- In the folder **SubRegion #0 > Modifiers**, create a **Moving body** modifier on the downloaded **Imported object**.

In the **Properties** window of the just created **Moving body**, specify:

Rotation

Rotation Speed [radian/s]

X	**= 0**	[radian s^{-1}]
Y	**= 0**	[radian s^{-1}]
Z	**= 600**	[radian s^{-1}]

Create the **Right rotor**:

- Select **Create** from the context menu of the **Objects** folder and create an **Imported object** by loading its geometry from the file `Compressor_Female.wrl`.
- In the folder **SubRegion #0 > Modifiers**, create a **Moving body** modifier on the downloaded **Imported object**.

In the **Properties** window of the just created **Moving body**, specify:

Mass Properties

Center of Inertia

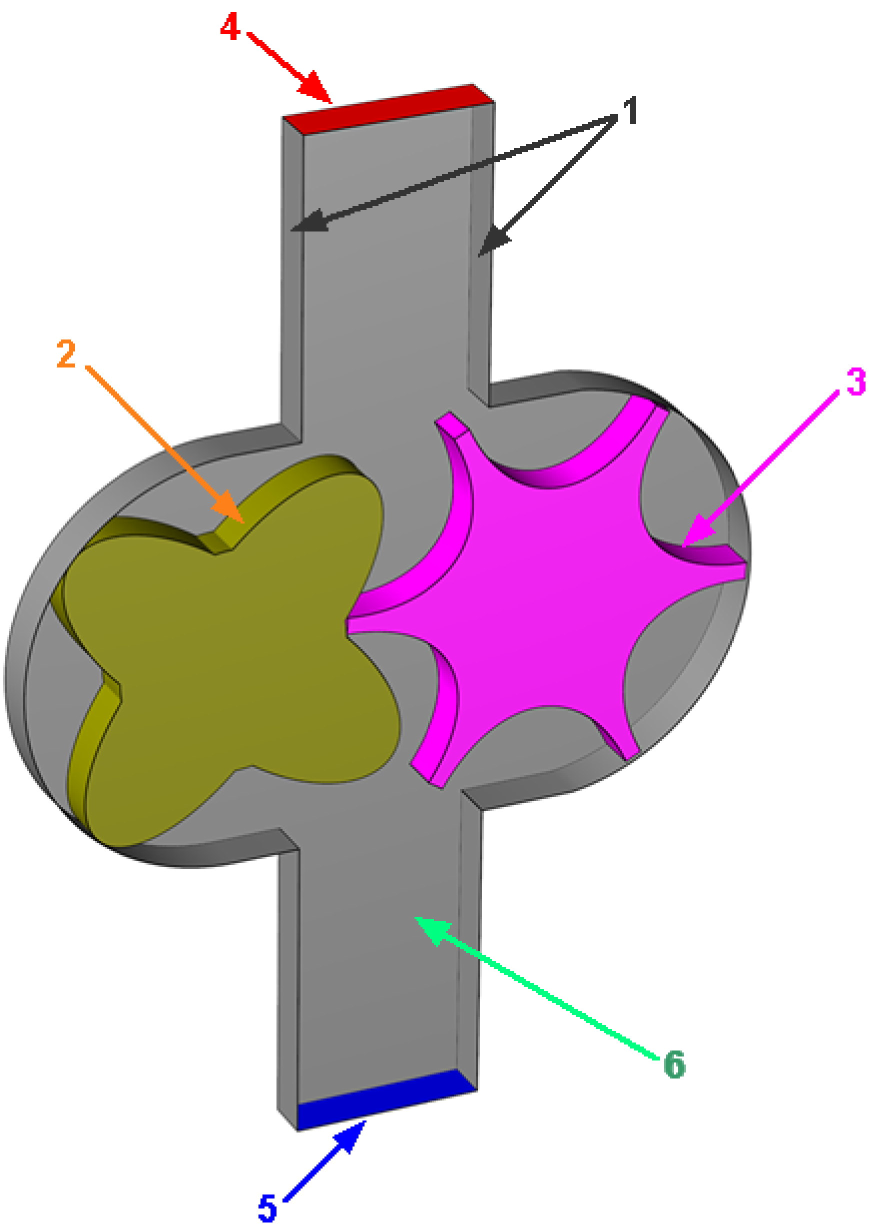
X	**= 0.085**	[m]
Y	**= 0**	[m]
Z	**= 0**	[m]

Rotation

Rotation Speed [radian/s]

X	**= 0**	[radian s^{-1}]
Y	**= 0**	[radian s^{-1}]
Z	**= -400**	[radian s^{-1}]

5.3.5.3 Boundary conditions



Specify the following boundary conditions:

Boundaries 1, 2, 3

Type **= Wall**

Variables

Temperature(Phase #0) = Zero gradient

Velocity(Phase #0) = Logarithm law

TurbEnergy(Phase #0) = Value in cell near wall

TurbDissipation(Phase #0) = Value in cell near wall

Boundary 4

Type = Inlet/Outlet

Variables

Temperature(Phase #0) = Total temperature

Value = 0 [K]

Velocity(Phase #0) = Total pressure

Total pressure = 0 [Pa]

TurbEnergy(Phase #0) = Pulsations

Value = 0

TurbDissipation(Phase #0) = Turbulent scale

Value = 0 [m]

Boundary 5

Type = Inlet/Outlet

Variables

Temperature(Phase #0) = Temperature

Value = 0 [K]

Velocity(Phase #0) = Total pressure

Total pressure = 1000 [Pa]

TurbEnergy(Phase #0) = Pulsations

Value = 0

TurbDissipation(Phase #0) = Turbulent scale

Value = 0 [m]

Boundary 6

Type = Symmetry

Variables

Temperature(Phase #0) = Symmetry

Velocity(Phase #0) = Slip

TurbEnergy(Phase #0) = Symmetry

TurbDissipation(Phase #0) = Symmetry

5.3.5.4 Initial conditions

In simulations of rotary compressors, it is recommended to specify in the area near outlet the same initial values of variables as the values on the outlet.

In **Model #0**, in **Init. data #0**, specify:

Pressure(Phase #0)

Value = 1000 [Pa]

In the folder **Objects**:

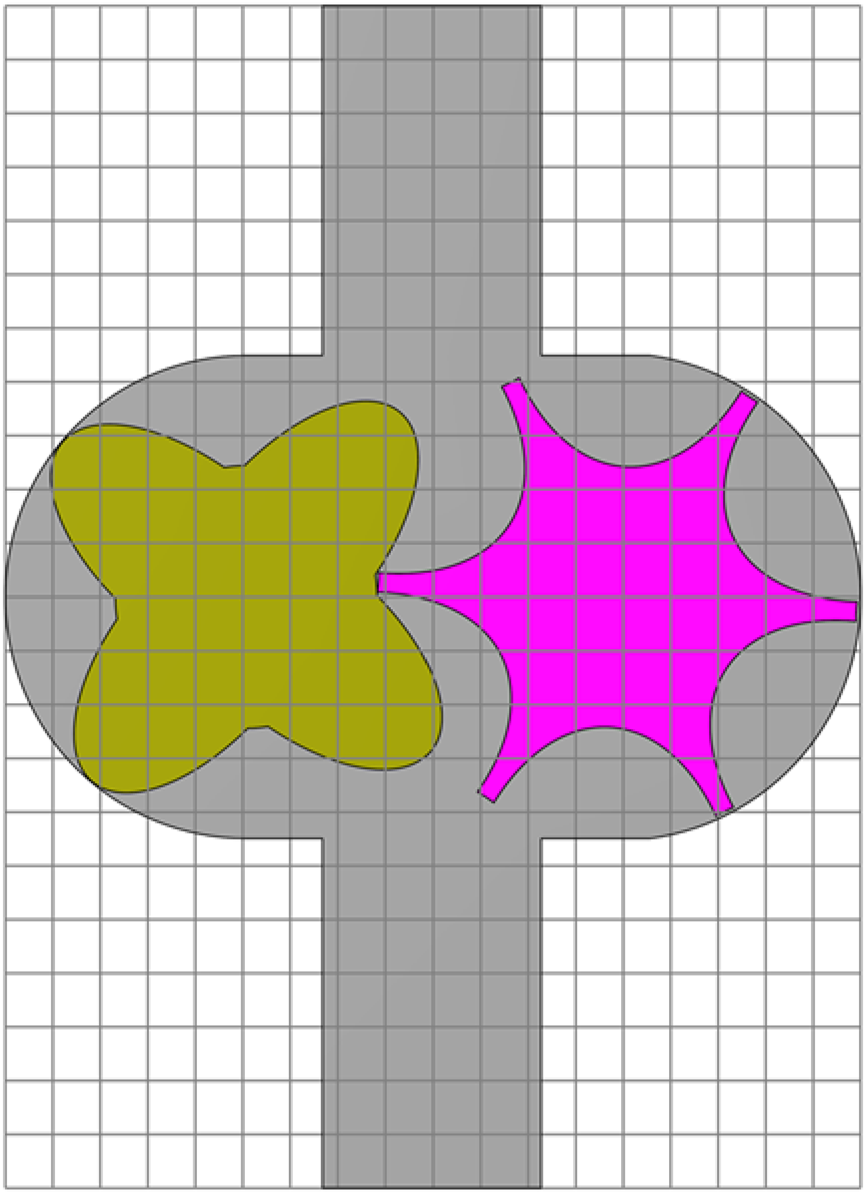
- Create **Box #0**.
- In the **Properties** window of **Box #0** specify:

Location			
Reference point			
X		= 0.04275	[m]
Y		= -0.095	[m]
Z		= 0.005	[m]
Size			
X		= 0.05	[m]
Y		= 0.08	[m]
Z		= 0.012	[m]

In **SubRegion #0**, in the **Properties** window of the element **Initial conditions > Init. condition #0**, specify:

Object	**= Box #0**
Init. data	**= Init. data #0**

5.3.5.5 Initial grid



Specify in the **Properties** window of the **Initial grid**:

Grid structure	**= 2D**
Plane	**= XY**
nX	**= 18**
nY	**= 22**

In the **Properties** window of the **Initial grid** click **Apply**.

5.3.5.6 Adaptation of the computational grid

Specify the adaptation in the area of gears.

Start with creation of a geometry object, within which the adaptation will act:

- Create **Box #1** in the folder **Objects**.
- Specify parameters in the **Properties** window of **Box #1**:

Location

Reference point

X	= 0.0427	[m]
Y	= 0	[m]
Z	= 0.005	[m]
Size		
X	= 0.2	[m]
Y	= 0.12	[m]
Z	= 0.012	[m]

Then specify the adaptation itself, which will act within **Box #1**:

- Create **Computational grid > Adaptation > Adaptation #0**.
- Add the geometry object **Box #1** to the subfolder **Computational grid > Adaptation > Adaptation #0 > Objects**.
- In properties of **Adaptation #0** specify:

Enabled	= Yes
Max level N	= 2
Layers	
Layers for Level N	= 4
Layers for Level N-1	= 4

5.3.5.7 History of the computation

In this simulation, it is supposed to display the history of the computation depending on time.

In order to display this history, it is necessary to:

- save the history of the computation
- create a layer for the visualization
- start sequential loading of the computation's results with saving images into files

5.3.5.8 Parameters of calculation

Specify in the **Solver** tab:

- In the **Properties** window of the **Data autosave** element, specify:

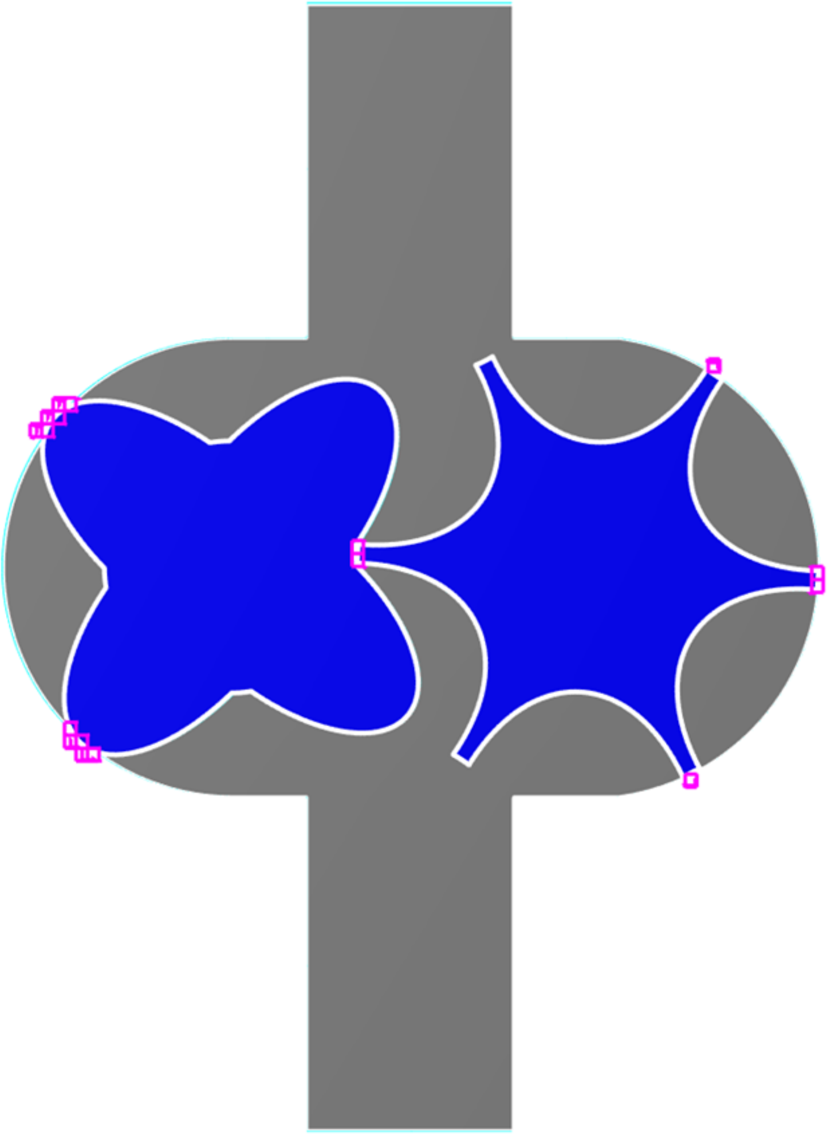
History	= Yes	
Frequency		
Type	= By time	
Number of seconds	= 0.0005	[s]

5.3.5.9 Visualization

To view the dynamics of the solution during the computation, prior to computation, specify visualization of:

1. **Distribution of gap cells**
2. Instant **Velocity distribution** in the plane of symmetry
3. **Dynamics of the velocity profile** depending on time

5.3.5.9.1 Distribution of gap cells



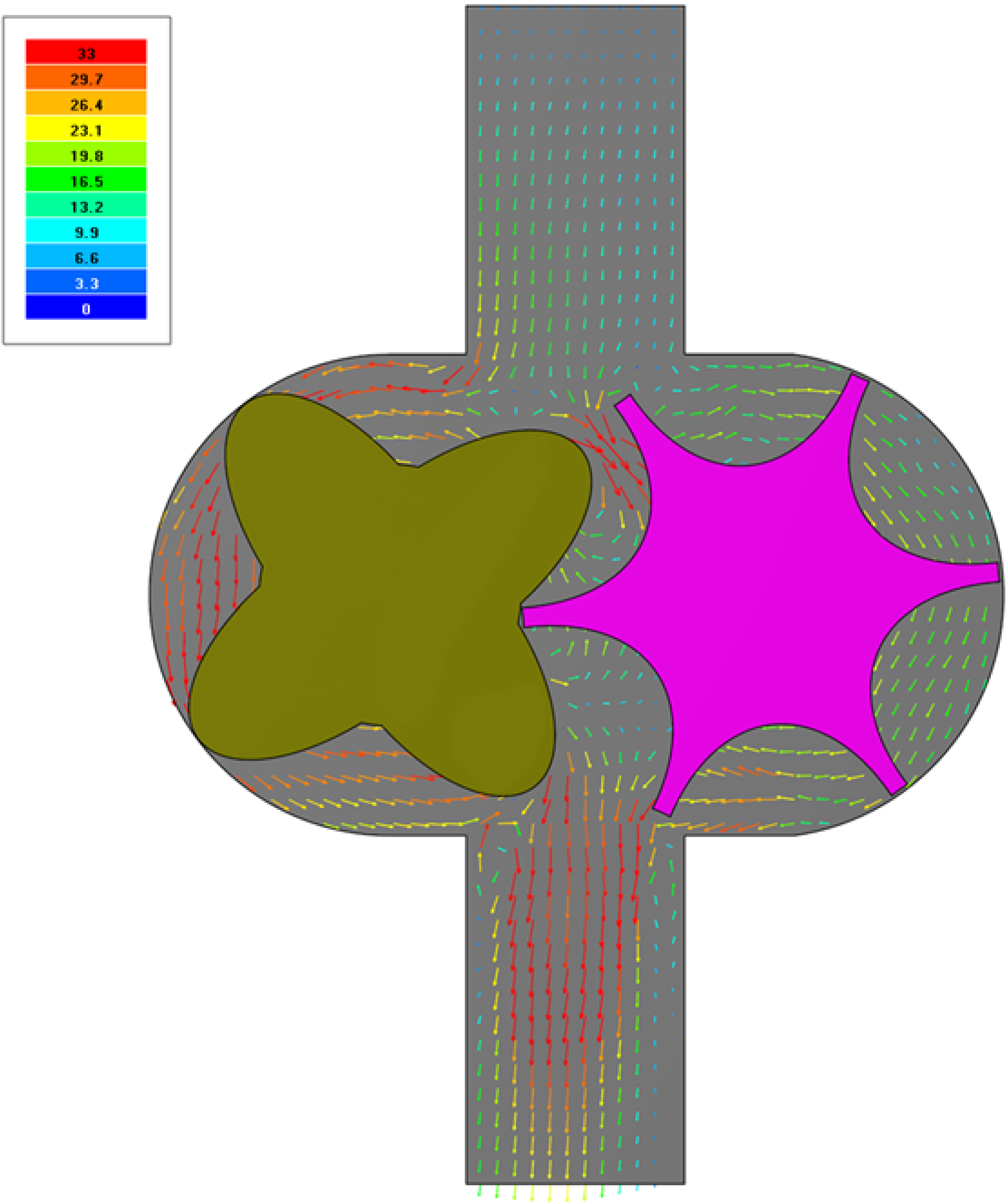
- Create a layer **Cell set** on the **Computational space**
- In the **Properties** window of this layer **Cell set**, specify:

Type	**= Gap**
Appearance	
Mode	**= Lines**

Note:

See section **Flow in clearance - use of the Gap model** for the algorithm of selecting gap cells.

5.3.5.9.2 Velocity distribution



- In the **Properties** window of **Plane #0**, specify:

Object

Normal

X	**= 0**
Y	**= 0**
Z	**= 1**

(to direct a **Plane**'s normal along the axis **Z**, you can also click the **Operations >** Z↓ button in the **Properties** window of the **Plane**)

- Create a layer **Vectors** on the **Plane #0.**
- In the **Properties** window of the layer, specify:

Grid

Size 1 **= 50**

Size 2 **= 50**

Coloring

Variable

Variable **= Velocity**

Value range

Mode **= Manual**

Max **= 33**

Min **= 0**

Palette

Appearance

Enabled **= Yes**

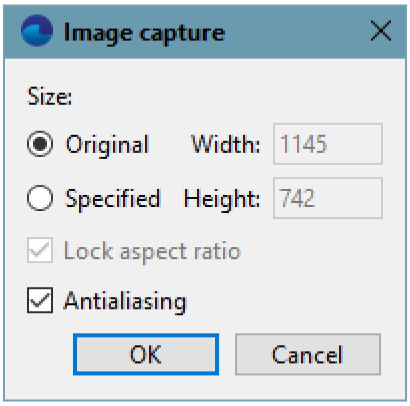
Style **= Style 1**

The program will automatically specify the variable, which is used to build the vectors, **Variable > Variable = Velocity**.

5.3.5.9.3 Velocity variation

To display changing of the velocity profile (field of velocities), do the following:

- Go to the first saved step by clicking the button (**Load first step**).
- Run sequential saving of the images from the **View** window into files by clicking the button (**Begin sequence**). The **Image capture** dialog box will open:



- Do not change sizes of the image that are set in the **Image capture** dialog box and click **OK**.
- In the dialog box for specifying files enter the prefix for file names of the image files (the program will create multiple files with names that will be formed from your prefix and sequential numbers).
- Run the sequential download of the data stored at intermediate steps by clicking the button (**Start playback**).
- After the sequential data download ends, stop the saving the images from the **View** window into files by clicking the button (**Finish sequence**).

5.3.5.9.4 Displaying a text in the View window

The text **Title** in the **View** window will display number of rotations of the left and the right rotors.

Create **Characteristics #0** on the left rotor using the **Create characteristics** command from the context menu of the **Imported object #0**, which corresponds to the left rotor. You can build **Characteristics #0** by any variable, this has no matter for results in this exercise.

Similarly create **Characteristics #1** on the right rotor.

Create a **User variable** corresponding to the number of revolutions of the left rotor. In **Preprocessor**, in the context menu of the **User variables > Global** apply the **Create > Scalar** command. In properties of the just created variable **UGV #0** specify:

Name = N_left
Value = abs(trunc(RotVelocity0_Z*Time/(2*PI)))

To specify **Value**, apply the **Formula editor**:

- **RotVelocity0_Z** is specified in the **Variables & constants** pane in the **Integral** tab, in the variable **Characteristics #0 (Imported object #0) > Rotation velocity > Z**.
- **Time** is specified in the **Variables & constants** pane in the **Integral** tab, in the variable **Internal characteristics > Current time**.
- **PI** is specified in the **Variables & constants** pane in the **Constants** tab, in the constant **Pi number**.

Similarly create a **User variable** named as **N_right** for number of revolutions of the right rotor.

Create **Stop criteria** in the folder **Stopping conditions > User values** in the **Solver** tab:

- Create **Stop criterion #0** and in its properties specify: **Object = N_left**.
- Create **Stop criterion #1** and in its properties specify: **Object = N_right**.

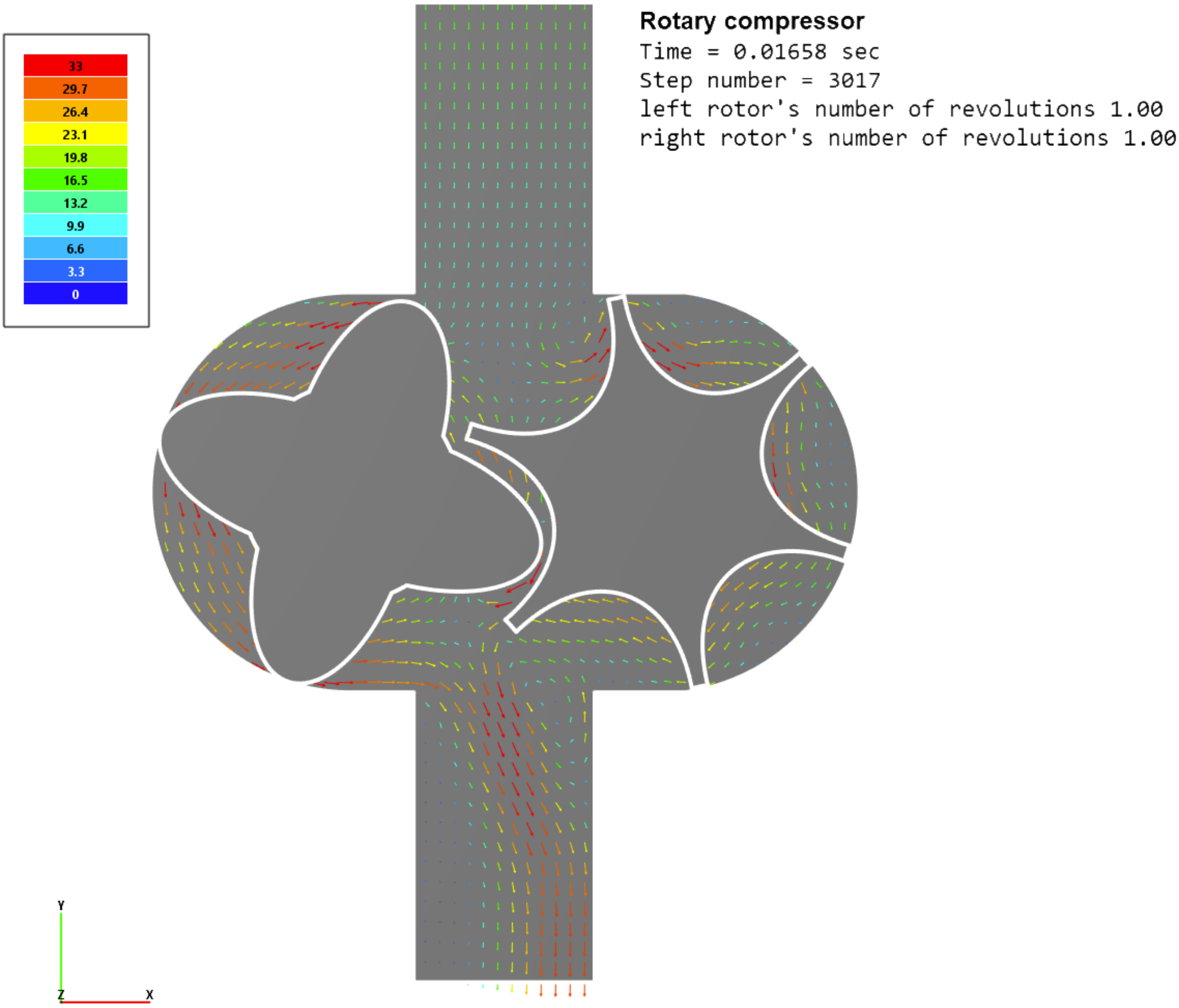
Define the text, which will be displayed in the **View** window. In the **Postprocessor** tab, in properties of the root element **3D-scene**, specify:

Show title = Yes
Title > Text = Rotary compressor
Title > Show time = Yes
Title > Show step number = Yes
Title > User values > [0] > Line begin = left rotor's number of revolutions
Title > User values > [0] > User stopper = Stop criterion #0
Title > User values > [1] > Line begin = right rotor's number of revolutions
Title > User values > [1] > User stopper = Stop criterion #1

Note: To make parameters **Title > User values > [0] > ...** and **Title > User values > [1] > ...** available, click twice the **User values >** **(Append item to the array)** screen button.

If necessary, change the text color of the title using the **Title > Text color** parameter so this text be not same as the background color of the **View** window.

The **View** window will display the text title with information about number of complete revolutions of the left and the right rotors.


33
29.7
26.4
23.1
19.8
16.5
13.2
9.9
6.6
3.3
0
Rotary compressor
Time = 0.01658 sec
Step number = 3017
left rotor's number of revolutions 1.00
right rotor's number of revolutions 1.00
Y
Z
X

5.4 Icing on a solid surface

In this case we consider simulating of icing on an airplane's surfaces. It is the problem, which has great practical value.

Conditions, when the icing is possible, are formed when an airplane flies in the troposphere in clouds or in super-cooled rain.

The icing is the most probable at temperatures from 0 °C to -20 °C. When the temperature is below -40 °C, the icing is unlikely.

The troposphere at temperatures below 0 °C contains, besides air, water vapor, waterdrops and ice crystals. Dispersed subcooled waterdrops are nuclei of condensation and crystallization of the water vapor.

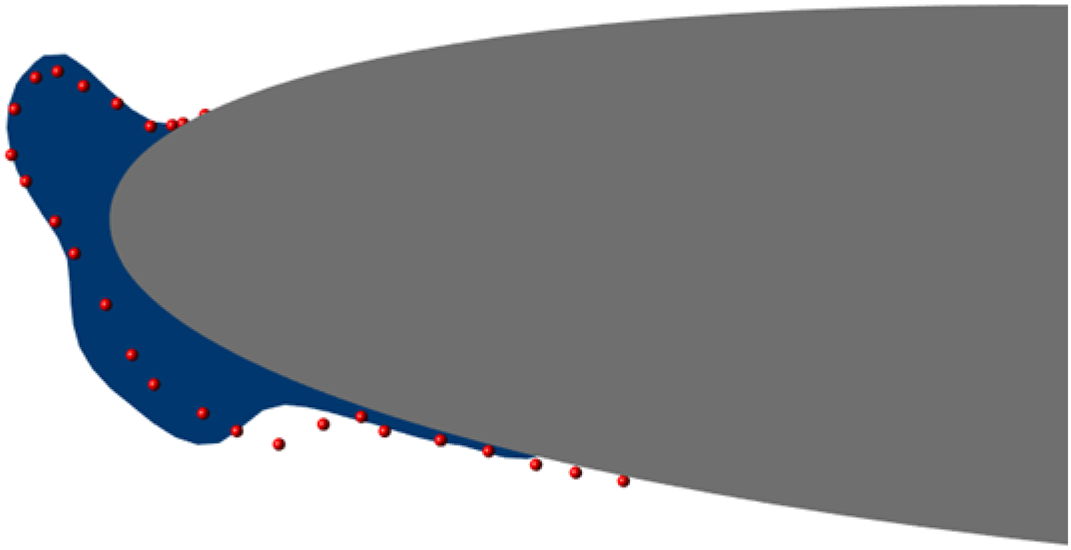
The main meteorological parameters, which influence on the icing rate, are:

- liquid water content (LWC), which measures the mass of condensed water containing in a unit volume of a cloud.
- temperature of the air (T)
- size of waterdrops (d)

FlowVision implements simulating of the following modes of icing:

- dry mode, when waterdrops impact the ice surface and freeze immediately
- wet mode, when subcooled waterdrops form a film on the solid surface. One part of this film becomes ice, while the other part, under the action of aerodynamic forces, flows along the external stream.

In this exercise we simulate the wet mode of icing; crystallization of subcooled waterdrops occurs on the surface of the *NACA0012* airfoil, which is being flown.



Parameters of the problem setting

Size and orientation:

chord length of *NACA0012*	c	= 0.5334	[m]
dimensions of the computational domain	5.3 × 5 × 0.00254		[m]
angle-of-attack	α	= 3.5	[degree]

Substances: air, liquid water, gas water (water vapor), solid water (ice)

Inlet parameters:

Pressure at infinity	p_∞	= 91201	[Pa]

Temperature at infinity	T_∞	= 262	[K]
Velocity on inlet (vector)	V	= (102.8, 0, 0)	[m/s]
Liquid water content (LWC)		= 1	[g/m^3]
Mean volume diameter (MVD) of particles in the dispersed phase		= 20 [μm] = 2×10^{-5} [m]	
Mass fraction of vapor in the air		= 0.0016	

Geometry:	`icing_plane.wrl` (for the computational domain) `icing_NACA0012.wrl` (for the airfoil)
Project:	`Icing_naca012`

Loading the geometry

Start your work from creation a new project based on the geometry `icing_plane.wrl`.

Then load an **Imported object** that corresponds to the airfoil. To do so, in the **Preprocessor** tab, open the context menu of the **Objects** folder, select there the **Batch import** and, in the operating system's dialog window, which opens, select the file `NACA0012_opt_airfoil.wrl`.

5.4.1 Physical model

In properties of the element **General settings** specify:

Reference values

Temperature	**= 262**	[K]
Pressure	**= 91201**	[Pa]

In the folder **Substances**:

- Create **Substance #0**.
- Load **Substance #0** from the **Standard** substance database:
  - From the context menu of **Substance #0** select **Load from SD > Standard**.
  - In the **Load from database** dialog box, which opens, select:

Substances	**= Air**
Phases	**= Gas (equilibrium)**

- Create three more **Substances** and load them from the **Standard** substance database. In their **Load from database** dialog boxes, which open, specify:
  - **Substances=Water**, **Phases=Liquid**
  - **Substances=Water**, **Phases=Gas (equilibrium)**
  - **Substances=Water**, **Phases=Solid (ice)**

In the folder **Phases**:

- Create a dispersed **Phase** of the **Particles** type (from the context menu of the folder **Phases** select the command **Create particles**). Rename this **Phase** as **WATERDROPS**.
- Add substance **Water_Liquid** into the folder **WATERDROPS > Substances**.
- In properties of the folder **WATERDROPS > Physical processes** specify:

Heat transfer	**= Convection & conduction**
Phase transfer	**= Convection & diffusion**
Motion	**= Motion**

Crystallization = Film model

- From the context menu of the folder **WATERDROPS > Size spectra** select the **Create** command. The element **WATERDROPS > Size spectra > Size spectrum #0** will appear in the project tree.
- Specify diameter of particles in the size group **[0]** of **Size spectrum #0**:

  Size groups > [0] > Diam. particles= 20e-6 [m]

- In properties of the element **WATERDROPS > Physical processes > Crystallization** specify:

Roughness model	**= Shin-Bond**	
LWC	**= 1**	[g/m^3]
Source smoothing	**= 1**	

- Create a continuous **Phase** and rename it as **ICE**.
- Add the substance **Water_Solid (ice)** into the folder **ICE > Substances**.
- In properties of the folder **ICE > Physical processes** specify **Heat transfer = Heat transfer via h**.
- In properties of the element **ICE > Physical processes > Heat transfer** specify **Time step coefficient = 1000000**. In this exercise the time step of the ice phase is substantially greater then the time step of the flow-around simulation. Value of the time step coefficient for the ice phase is several orders of magnitude greater than for other processes to obtain the temperature equilibrium in the ice phase be settled at one time step of the flow-around simulation.
- Create a continuous **Phase** and rename it as **AIR**.
- Add, in sequential order, substances **Water_Gas (equilibrium)** and **Air_Gas (equilibrium)** into the folder **AIR > Substances**.
- In properties of the folder **AIR > Physical processes** specify:

Heat transfer	**= Heat transfer via H**
Motion	**= Navier-Stokes model**
Mass transfer	**= Mixing**
Turbulence	**= KES**

- In properties of the element **AIR > Physical processes > Heat transfer** specify **All terms = Yes**. This setting enables taking into account the heat generation due to viscous dissipation.
- In properties of the element **AIR > Physical processes > Motion** specify **Visc. force supplement = Yes**. Solving the full equation allows the program to obtain heat flows that determine more realistic shape of the ice body.
- In properties of the element **AIR > Physical processes > Turbulence** specify **Roughness constant = 0.097**. The value of the **Roughness constant** is tried individually for each computational case. This value depends on geometry of the object, Y+, and LWC.

In the folder **Models**:

- Create **Model #0**.
- Sequentially (one by one) add phases **ICE**, **AIR**, and **WATERDROPS** into folder **Model #0 > Phases**.
- In properties of the element **Model #0 > Phase interaction > Continuum-particles**, which has **Phase0=ICE** and **Phase1=WATERDROPS**, specify:

  Is carrier phase = No

- In properties of the element **Model #0 > Phase interaction > Continuum-particles**, which has **Phase0=AIR** and **Phase1=WATERDROPS**, specify:

Substance pair > [0] > Phase0	**= Water_Gas (equilibrium)**
Substance pair > [0] > Phase1	**= Water_Liquid**
Cd	**= Model1**
Nu	**= Model1**

- For the element **Init. data > Init. data #0** specify:

Velocity(AIR)

Value > X	**= 102.8**	[m/s]

Mass frac. [Water_Gas (equilibrium)](AIR)

Value	**= 0.0016**

VOF(AIR)

Value	**= 1**

Phase volume(WATERDROPS)

Value	**= 1e-6**

Velocity (disp.)(WATERDROPS)

Value > X	**= 102.8**	[m/s]

In properties of **SubRegion #0**, specify **Model = Model #0**.

5.4.2 Moving body

Add to the project a **Moving Body** modifier that will correspond to the airfoil and set its properties. This **Moving Body** is to be built on the **Imported object #0**, which was added into the project before (see "Loading the geometry and stages of the computation"). Follow these steps:

- In the folder **SubRegion #0 > Modifiers** create a **Moving Body** modifier on the loaded **Imported object #0**.
- Rotate the new **Moving Body #0** by the angle-of-attack α=5º. To do so, in properties of **Moving Body #0**, click the button **Initial position > Operations >** (**Relative rotation around local axis Z**) and, in the dialog box, which opens, set the rotation angle as **-3.5** degrees (a negative value).
- In properties of **Moving Body #0** click **Apply** and then click the screen button **Operations >** **(Place to initial position)**.
- In properties of **Moving Body #0** specify:

Activation

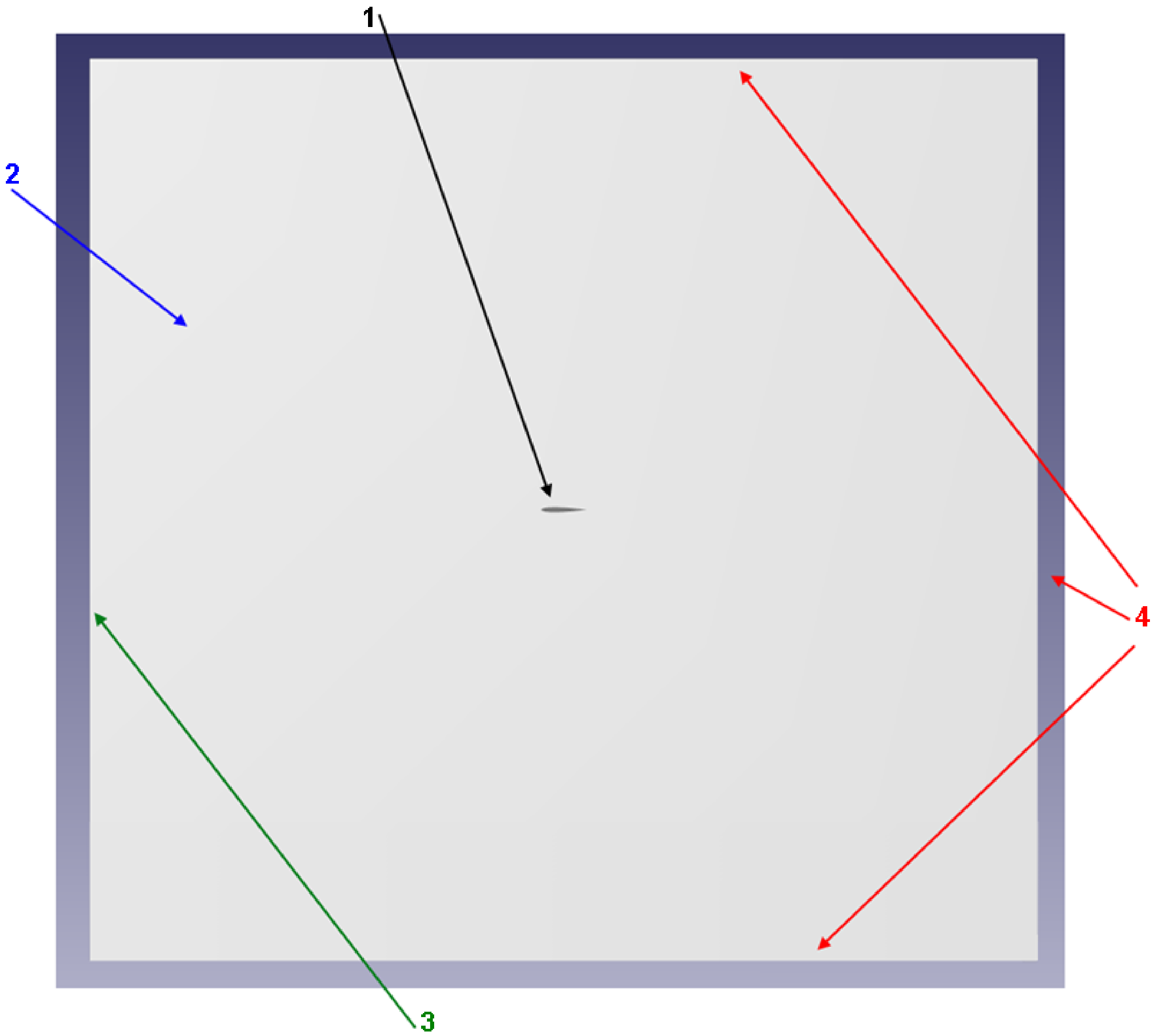
Type	**= Permanent**

Update

Type	**= Disabled**

(position and orientation of the wing, on which the icing will be simulated, will not change, so we disable update of **Moving Body #0**)

5.4.3 Boundary conditions



Specify boundary conditions according to the illustration:

Boundary 1

Name	**= Wing**
Type	**= Wall, film**
Variables	
VOF(ICE)	**= Value**
Value	**= 1**
VOF(AIR)	**= Value**
Value	**= 0**

Boundary 2

Name	**= Symmetry**
Type	**= Symmetry**

Boundary 3

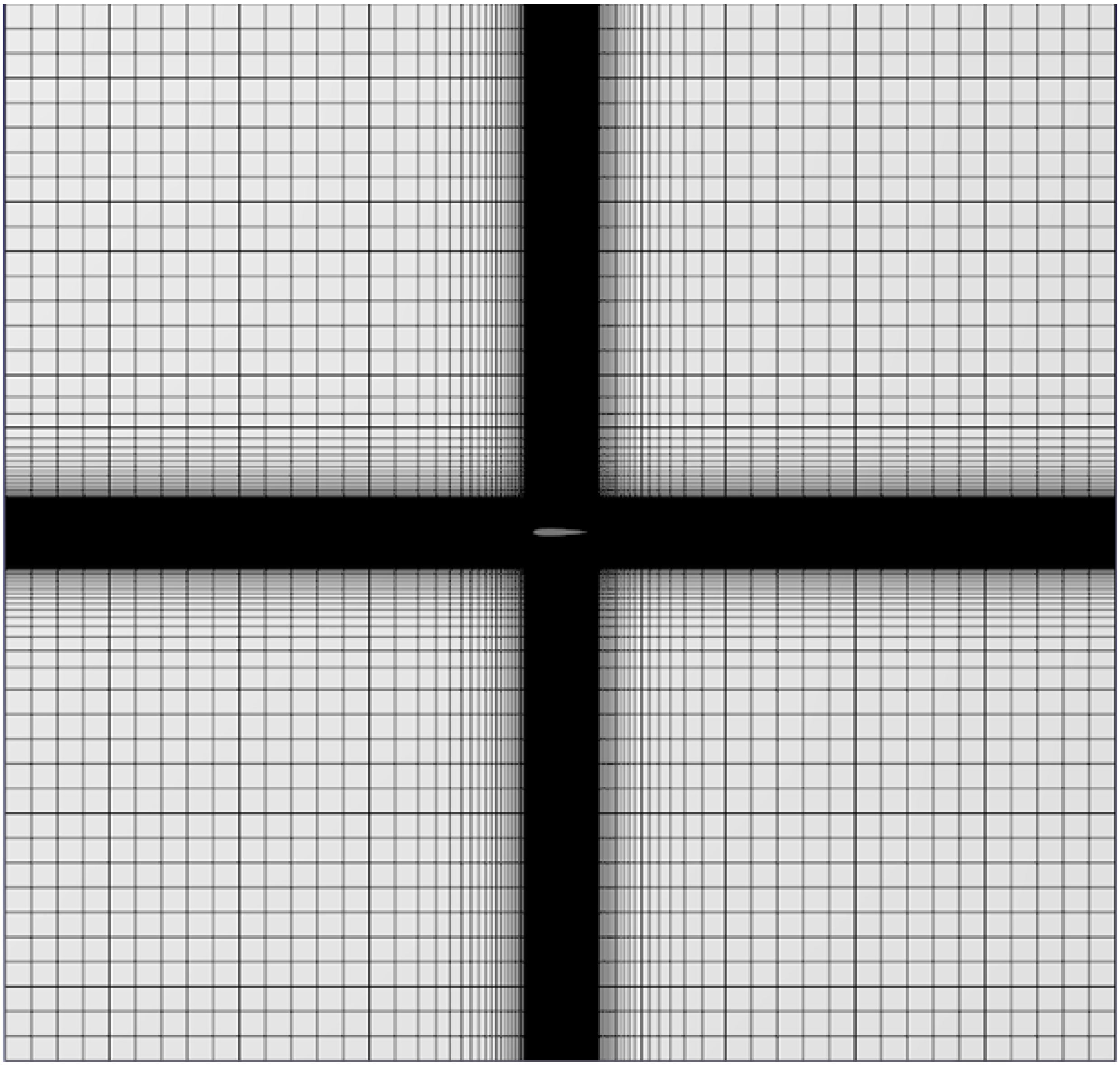
Name	**= Inlet**	
Type	**= Inlet/Outlet**	
Variables		
Temperature(ICE)	**= Temperature**	
Value	**= 0**	[K]
VOF(ICE)	**= Value**	
Value	**= 0**	
Temperature(AIR)	**= Temperature**	
Value	**= 0**	[K]
Velocity(AIR)	**= Velocity with pressure**	
Velocity > X	**= 102.8**	[m/s]
Velocity > Y	**= 0**	[m/s]
Velocity > Z	**= 0**	[m/s]
Pressure	**= 0**	[Pa]
Mass frac. [Water_Gas (equilibrium)](AIR)	**= Value at the inlet**	
Value	**= 0.0016**	
TurbEnergy(AIR)	**= Pulsations**	
Value	**= 0**	
TurbDissipation(AIR)	**= Turbulent scale**	
Value	**= 0**	[m]
VOF(AIR)	**= Value**	
Value	**= 1**	
Phase volume(WATERDROPS)	**= Value**	
Value	**= 1e-6**	
Temperature (disp.)(WATERDROPS)	**= Value**	
Value	**= 0**	[K]
Velocity (disp.)(WATERDROPS)	**= Particles velocity**	
Particles velocity > X	**=102.8**	[m/s]
Particles velocity > Y	**= 0**	[m/s]
Particles velocity > Z	**= 0**	[m/s]

Boundary 4

Name	**= Outlet**	
Type	**= Free outlet**	
Variables		
Temperature(ICE)	**= Temperature**	
Value	**= 0**	[K]
VOF(ICE)	**= Zero gradient**	
Temperature(AIR)	**= Zero gradient**	
Velocity(AIR	**=Pressure**	
Value	**= 0**	[Pa]
Mass frac. [Water_Gas (equilibrium)](AIR)	**= Zero gradient**	
TurbEnergy(AIR)	**= Zero gradient**	

TurbDissipation(AIR)	**= Zero gradient**
VOF(AIR)	**= Zero gradient**
Phase volume(WATERDROPS)	**= Permeable surface**
Temperature (disp.)(WATERDROPS)	**= Permeable surface**
Velocity (disp.)(WATERDROPS)	**= Permeable surface**

5.4.4 Initial grid



In properties of the **Initial grid**, click the button  to open the **Initial grid editor**.

Specify in the **Initial grid editor**:

for axis OX

Grid parameters

h_max	**= 0.5**	[m]
h_min	**= 0.0025**	[m]

Insert a reference line with coordinate **x=0** [m].

Specify **Reference line parameters** for the reference line with coordinate **x=0**:

h	**= 0.0025**	[m]
kh-	**= 1**	

kh+ **= 1**

for axis OY (click the button **Y**)

Specify the same parameters as for axis OX.

Click **OK** to close the **Initial grid editor** with saving the entered data.

As in this exercise the problem is simulated in 2D setting, there will be only one cell along axis OZ. Specify in properties of the **Initial grid**:

Grid structure = 2D

Plane = XY

In the **Properties** window of the **Initial grid** click **Apply**.

5.4.5 Parameters of calculation

Specify in the **Solver** tab:

- In properties of the **Time step** element, specify:

Method	**= Via CFL number**
Convective CFL	**= if(PhaseTime<231;10;-1)**
Surface CFL	**= 1**
Film CFL	**= 5**

The computation will automatically stop after 231 seconds of icing.

- In properties of the **Advanced settings** element, specify:
    - **Multiphase C > Phase conservative = No**. The mass conservation law effects not strictly at motion of the boundary of the ice body. This is admissible because the flow is simulated in non-closed space.
    - **Multiphase C > CFL for VOF source = 0.1**
    - **Multiphase C > Use for time step = No**. The time step is calculated without taking into account the velocity of the inter-phase surface.
    - **Multiphase D > Cloud boundary = 0**.
    - **Multiphase D > Activation of disp. phase crystallization > Type = Start in steps**
    - **Multiphase D > Activation of disp. phase crystallization > Start in steps = 401**
    - **Turbulence > WF: profile T+ = 1**. This parameter specifies the temperature profiles, which is used in the computation.
    - **Smooth diff. fluxes = Yes**. This setting enables smoothing the values of **Heat flux** on solid surfaces.

- In properties of the element **Limiters > Limiters for calculation > Phase Limiters > AIR**, specify:

Limiter

Density, min.	**= 0.01**	[kg m^{-3}]
Temperature abs, min.	**= 25**	[K]
Temperature abs, max.	**= 550**	[K]
Velocity, max.	**= 1000**	[m/s]
Pressure abs, min.	**= 1000**	[Pa]

- In properties of the element **Limiters > Limiters for calculation > Phase Limiters > ICE**, specify:

Small Cells

Criterion **= Relative**

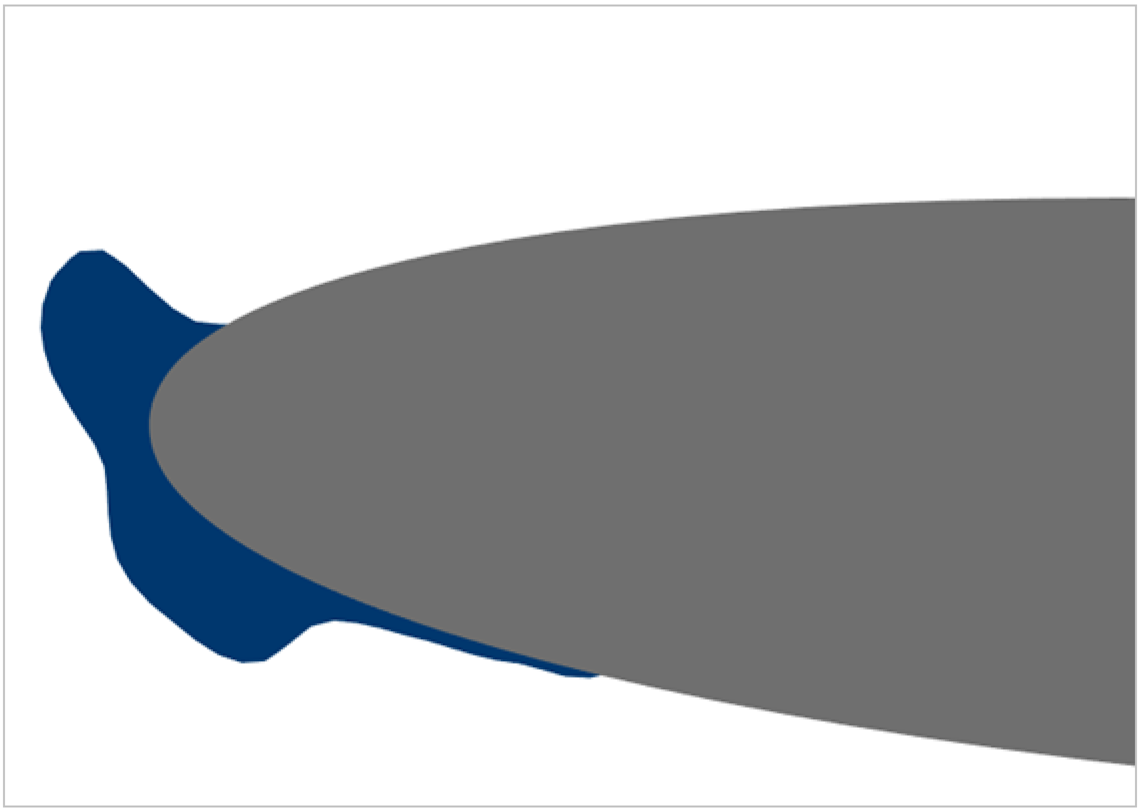
When icing is simulated, it is recommended to use the **Relative** criterion for revealing small cells for the solid-state **Phase** (ice body); this is set by the **Small Cells > Criterion** parameter in properties of the element **Limiters > Limiters for calculation > Phase Limiters > Phase #N** in the project tree.

5.4.6 Visualizing results of the computation

Shape of the ice body

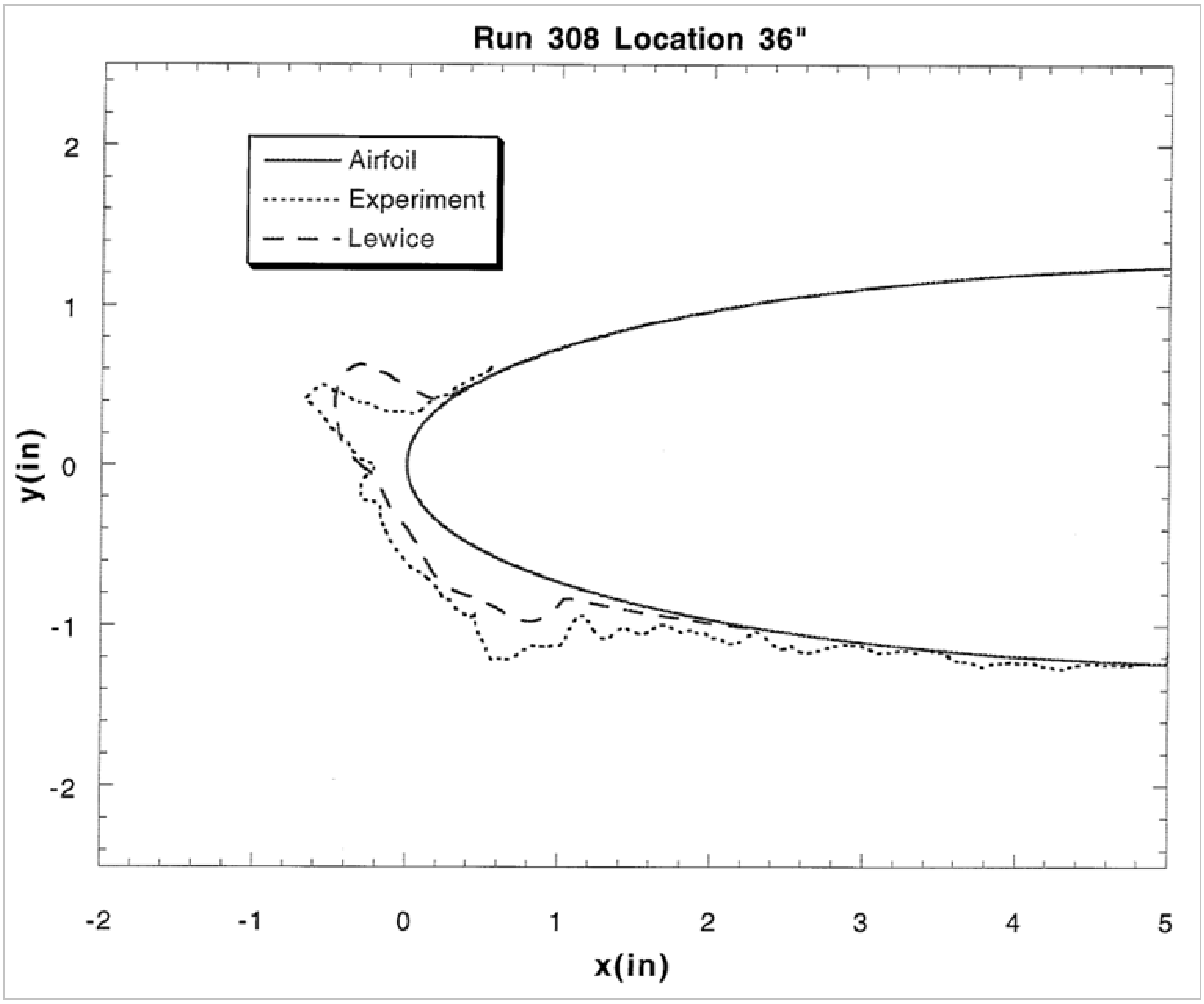
In the **Postprocessor** tab, on the object **Computational space**, create a new layer **VOF #0** and specify in its properties **Phase = ICE**.

Shape of the ice body at the moment when the computation stops (at the 231-rd second of the icing) is shown on the illustration below:

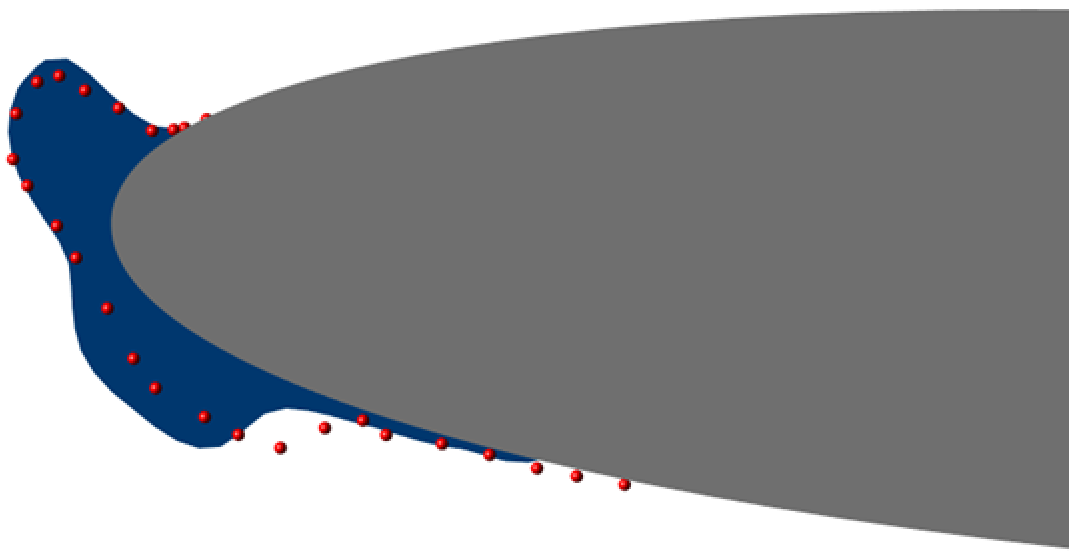


Comparing to the ice shape from the experiment

Conditions of the icing simulating in this project correspond to experiments in the wind tunnel *NASA Glenn Icing Research Tunnel (IRT)* [Wright, Rutkowski, 1999]: run 308.



In the **Postprocessor** tab create a **Set of sensors** object based on the file `Geom/run308_ice`. To do so, apply the **Operations > Overwrite sensors from file** button in properties of the **Set of sensors**. This allows you to visually compare results of the simulating and the experimental data:



6 Coupling with other software

6.1 Deformable valve in channel

Fluid-structure interaction (FSI) is interaction of motion or deforming body and internal or external flow. This kind of analysis is important when you develop engineering systems and solve problems in areas of aircraft engineering, design of motors, bridges, etc.

Problems of interaction between a constructions and a flow commonly are too difficult for analytical approach, and these problems can be analyzed in an experiment or by numerical simulation. For numerical simulations of FSI problems, two approaches can be used:

- jointless approach (equations for flow and equations for deformations are solved simultaneously by a single solver);
- concurrent simulation (equations for flow and equations for deformations are solved by different programs).

One of features of *FlowVision* is possibility of coupled computations with external programs. The current version of *FlowVision* allows doing FSI computations with the *Abaqus* software package via **Pre-Postprocessor**, which is being connected to **Solver**.

A correct start of joint computation requires manual modification of *Abaqus*' and *FlowVision*'s files. Connection between these programs is impossible without these modifications.

A flow of liquid is partially overlapped by a valve, which obstructs the flow. A joint computation in *Abaqus* and *FlowVision* simulates the interdependent movement of the valve and parameters of the liquid's flow.

Parameters of the problem setting

Inflow parameters:			
Velocity	V	= 10	[m s^{-1}]
Fluid parameters:			
Density	ρ	= 1000	[kg m^{-3}]
Viscosity	μ	= 0.001	[kg $m^{-1}s^{-1}$]
Parameters of the valve:			
Density	ρ	= 3500	[kg m^{-3}]
Young's modulus	E	= 5×10^9	[Pa]
Poisson's ratio	ν	= 0.3	

Files

Geometry model of the tube	`Valve_Channel.wrl`
Project in *Abaqus*	`OneValve.inp`
Project in *FlowVision*	`Valve_Channel`

6.1.1 Preparing the project in Abaqus

In *Abaqus* deformations of the valve are simulated according to the loads that are received from *FlowVision*.

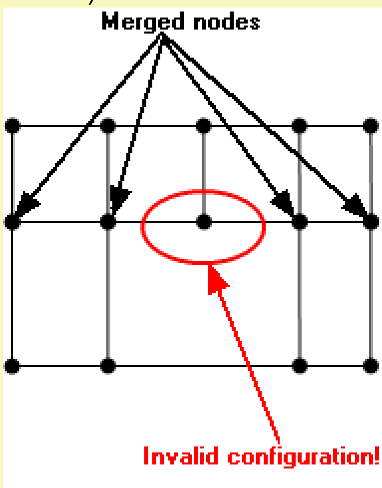
In this exercise a ready *Abaqus* model can be used (file `OneValve.inp`).

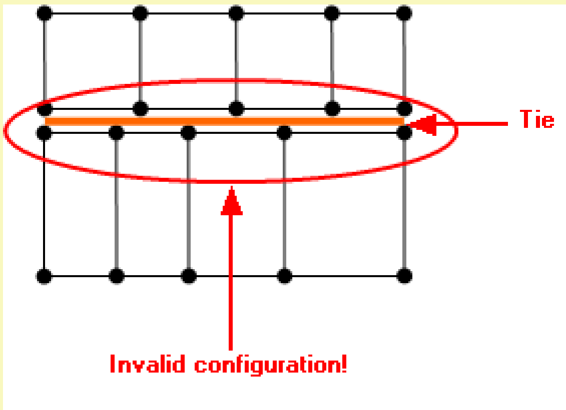
To obtain experience of creation an *Abaqus* model, see descriptions in sections:

- Creation a geometry model in Abaqus
- Specifying an interface surface in Abaqus
- Specifying boundary conditions and loads in Abaqus
- Generating an inp-file
- Modifying the inp-file of the Abaqus' project

Here are the requirements to the geometry model in *Abaqus*:

1. Details of the model involved in the coupling analysis should be simulated using volume or flat elements.
2. In the model interface surface(s) (which are interface regions of the **SURFACE** type) for the coupling analysis (the direct coupling interface) must be defined.
3. An interface surface must be:
   - closed
   - determined on a continuous mesh. The mesh must have no "duplicate" nodes, modified elements and must not contain a surface of two meshes connected with TIE-contact (see the illustration)





6.1.1.1 Creating a geometry model in Abaqus

This section helps you to create the valve's model in *Abaqus* (version *2017*) by your own. You can skip this section if you use a ready model `OneValve.inp` and return to the section when you wish to receive appropriate experience.

Follow the steps:

1. Click the icon  (**Create Part**). The **Create Part** dialog box will open, specify there:

    Name = Valve

    Modeling space = 3D

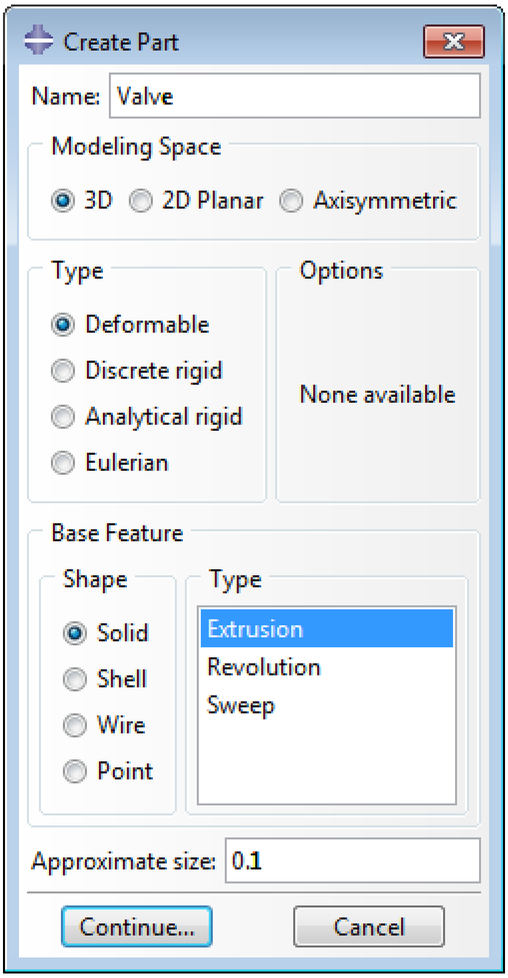
    Type = Deformable

    Base Feature > Shape = Solid

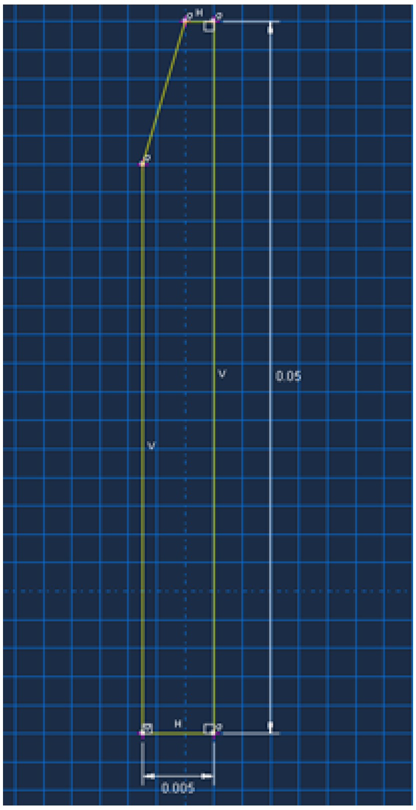
    Base Feature > Type = Extrusion

    Approximate size = 0.1

   and click **Continue**.

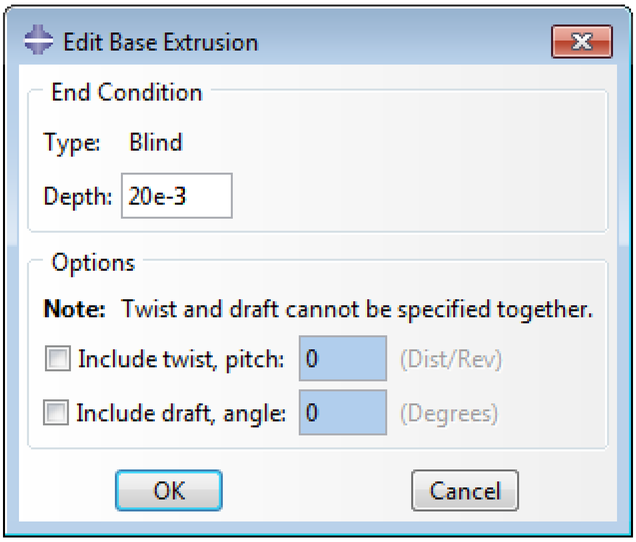


2. In the graphical work window, which opens, use the tool  (**Create Isolated Point**) to create points, which will be used to create the valve's contour. To do so, enter the following coordinates: **[-0.003,-0.01];[0.002,-0.01];[0.002,0.04];[0,0.04];[-0.003,0.03]**. Then use the tool  (**Create Lines Connected**) to outline the contour by lines. To do so, click **Done** (the mouse wheel). ***Important note:*** the created contour must be closed.

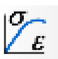


Making a contour for the part Valve

3. The **Edit Base Extrusion** dialog box will open. Enter the value **20e-3** into the field **Depth** and click **OK**:



Creating the part Valve

4. Navigate to the module **Property**. Click  (**Create Material**) to create a new material.
5. The **Edit Material** dialog box will open; specify there:

   Name = Metal (name of the material)

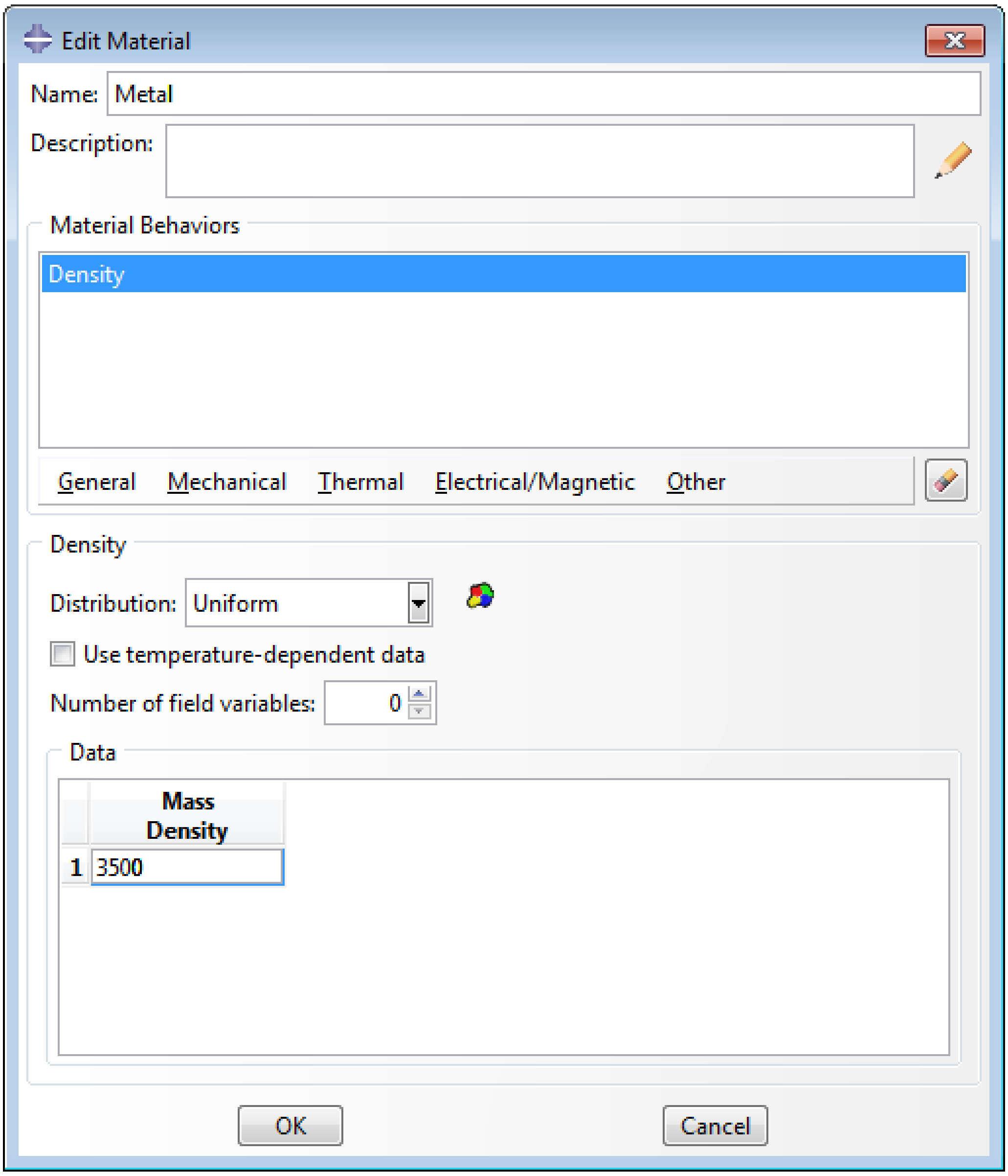
   General > Density = 3500 (density of the material, [kg·m^{-3}])

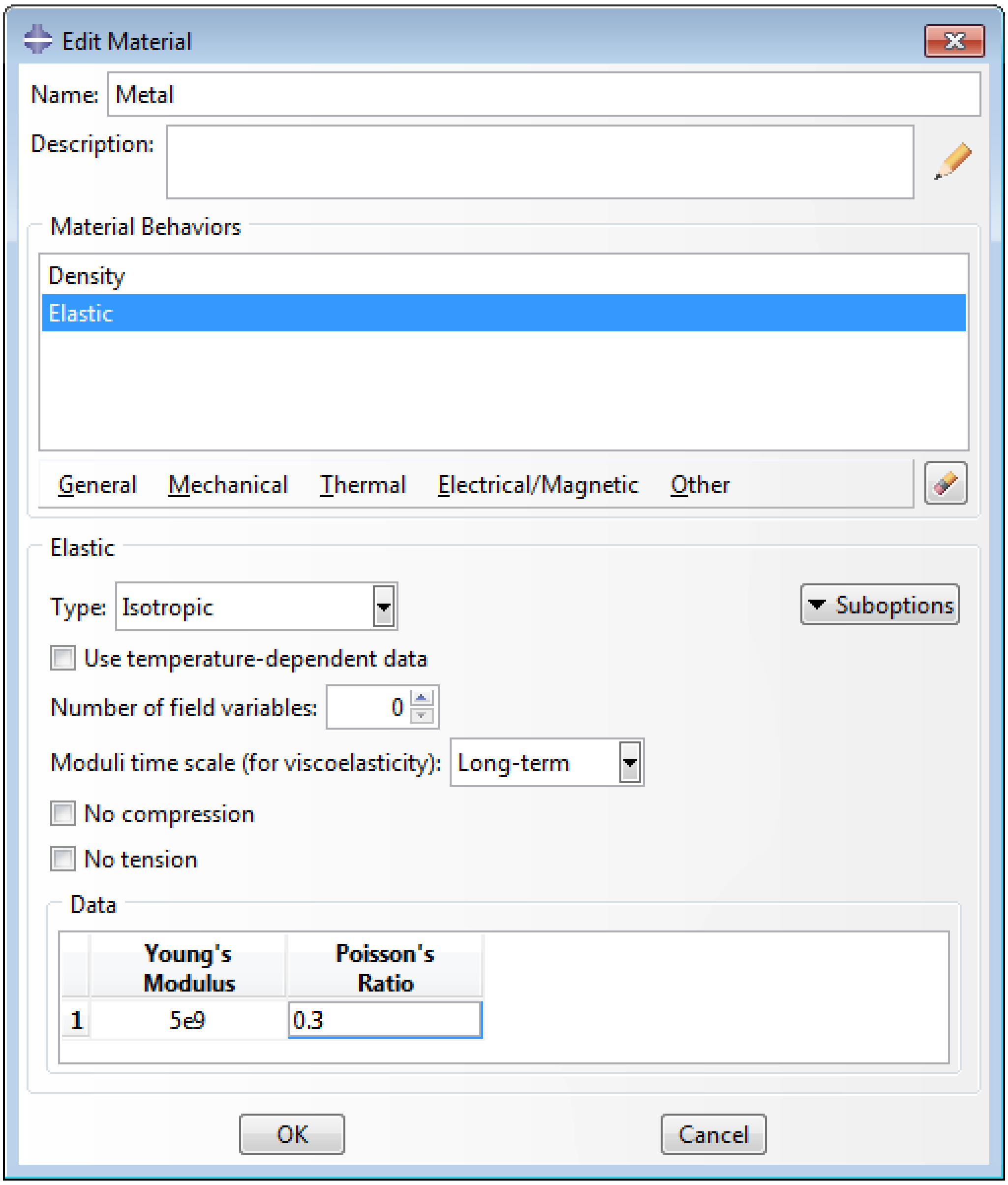
   Mechanical > Elasticity = Elastic

   Mechanical > Young's Modulus = 5E9 (Young's modulus, [Pa])

   Mechanical > Poisson's Ratio = 0.3 (Poisson's ratio)

   and click **OK**.


Edit Material
Name: Metal
Description:
Material Behaviors
Density
General Mechanical Thermal Electrical/Magnetic Other
Density
Distribution: Uniform
Use temperature-dependent data
Number of field variables: 0
Data
Mass Density
1 3500
OK
Cancel



Specifying properties of the material

6. Create a section made of the material **Metal**. Click  to open the **Create Section** dialog box. Specify there:

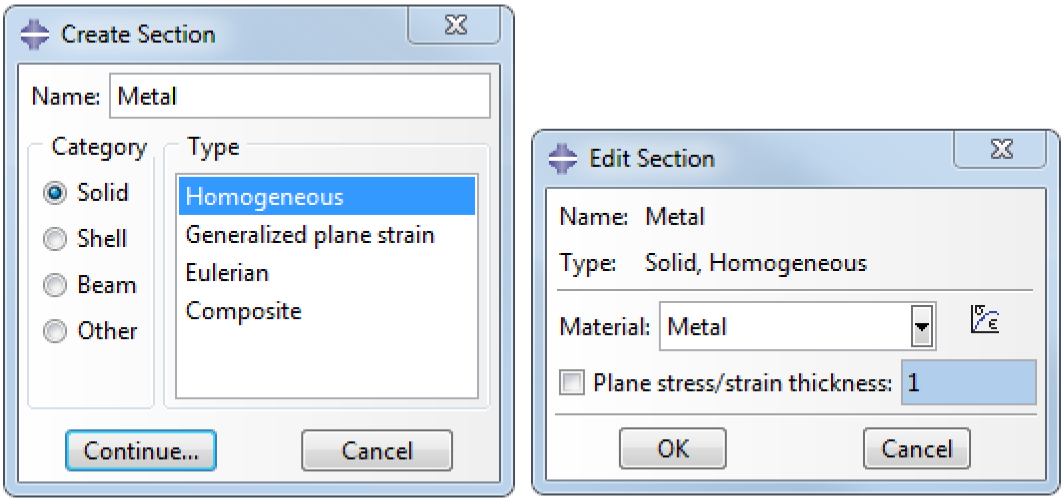
Name = Metal (name of the section)

Category = Solid

Type = Homogeneous

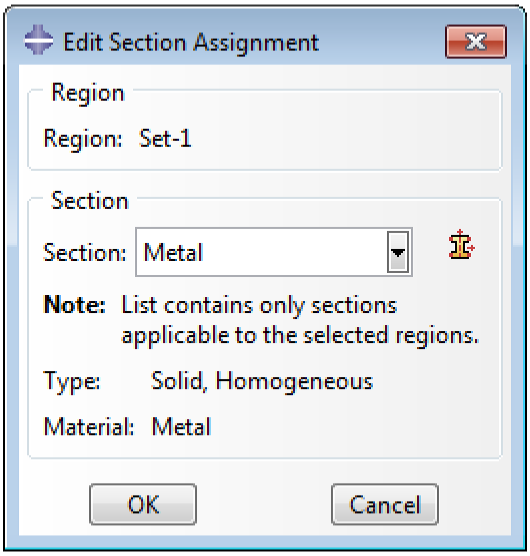
and click **Continue**.

The **Edit Section** dialog box will open. Select there **Material = Metal** and click **OK**.



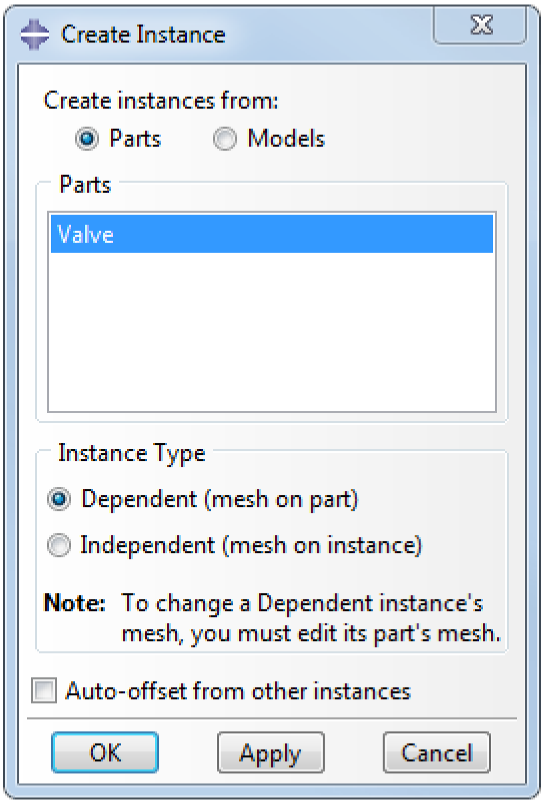
Creating and editing a section

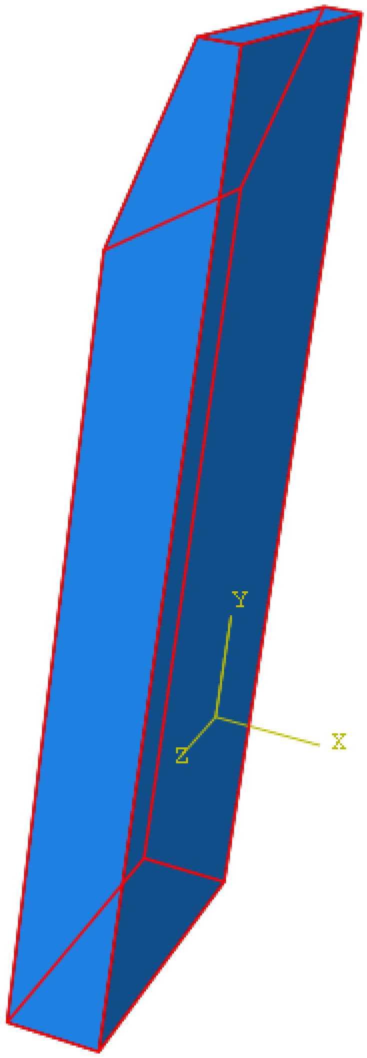
7. Click  (**Assign Section**) and in the dialog box **Edit Section Assignment** assign the just created section to the part, which you have created. Select the whole valve (it will be highlighted by red) and click **Done**. In the dialog box, which opens, click **OK**.

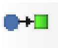


Assigning a section to the part Valve

8. Navigate to the module **Assembly**. To create an instance, click  (**Create Instance**). A window will open with a list of created parts. Select the part **Valve** and specify **Instance type = Dependent**. The just created part will appear in the graphical work window. Click **OK**.



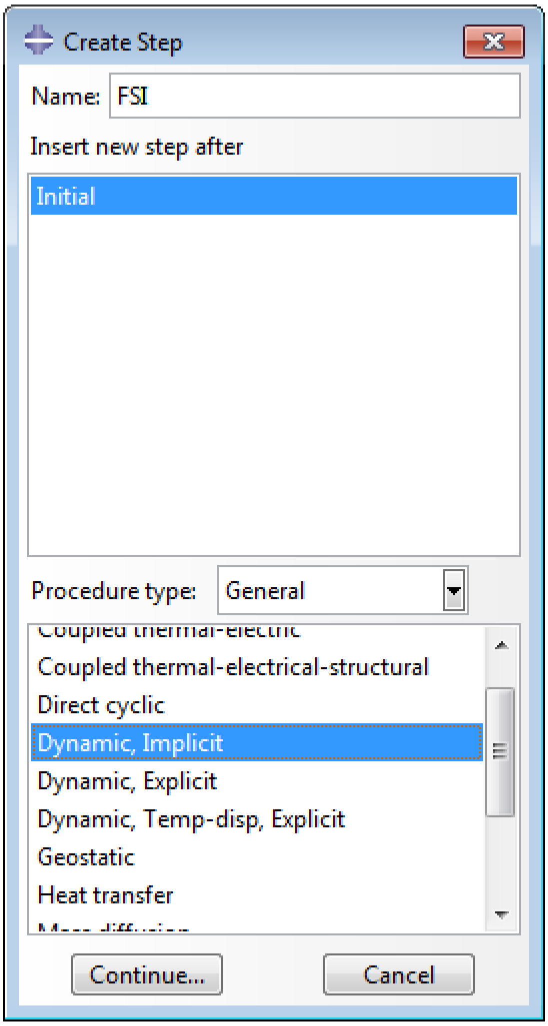


9. Specify the procedure of analysis. The numerical simulation will consist of one step (co-simulation). Navigate to the module **Step**, click  (**Create Step**). In the dialog box **Create Step** specify:

Name = FSI

Procedure Type = General

From the list below select **Dynamic, Implicit** and click **Continue**.

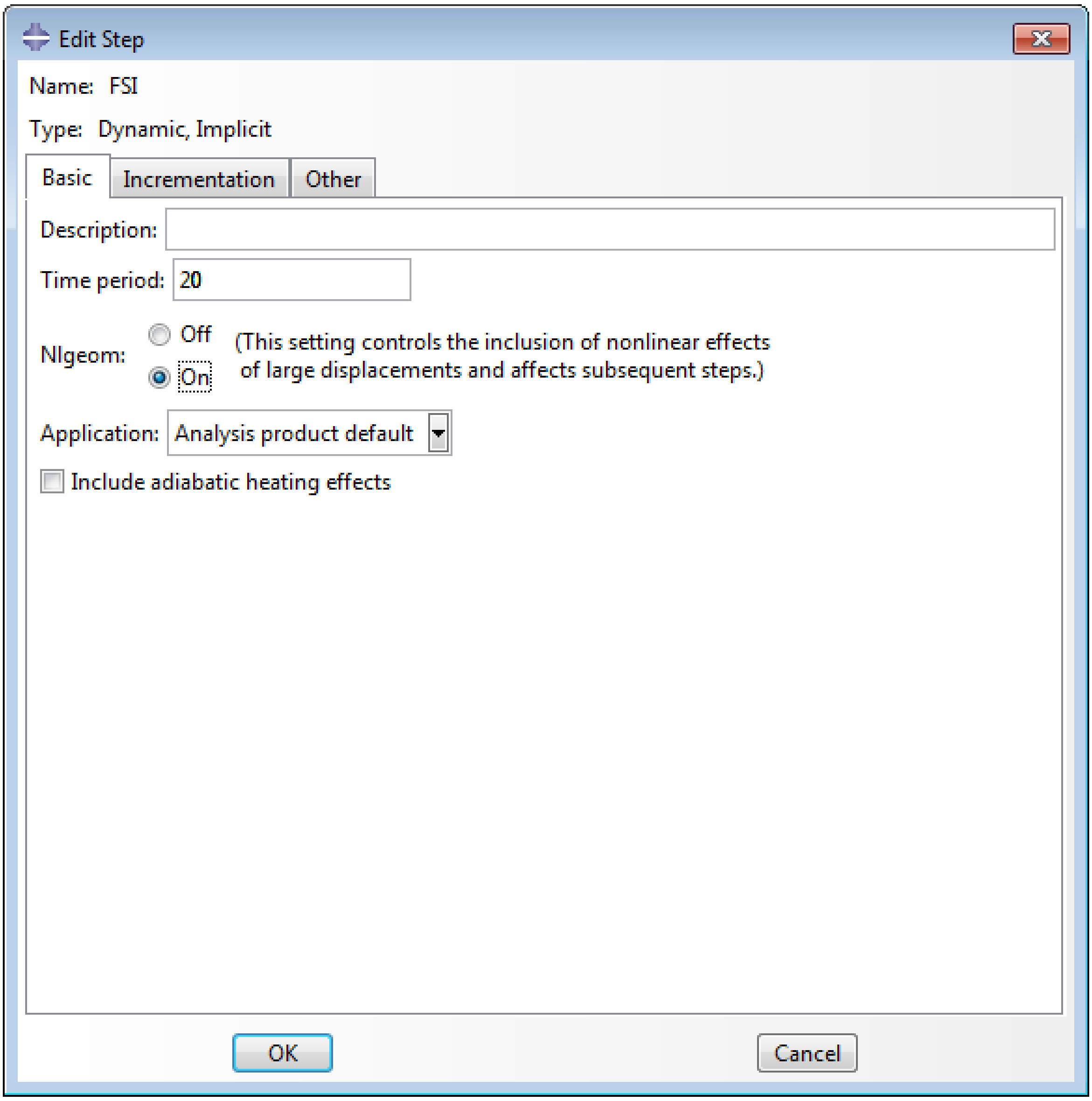


Specifying the procedure of analysis. Creation the FSI step. (1)

10. The **Edit Step** dialog box will open where you have to specify parameters of the step, which will be used in the analysis. In the **Basic** tab specify:

Time Period = 20

Nlgeom = On



Specifying the procedure of analysis. Creation the FSI step. (2)

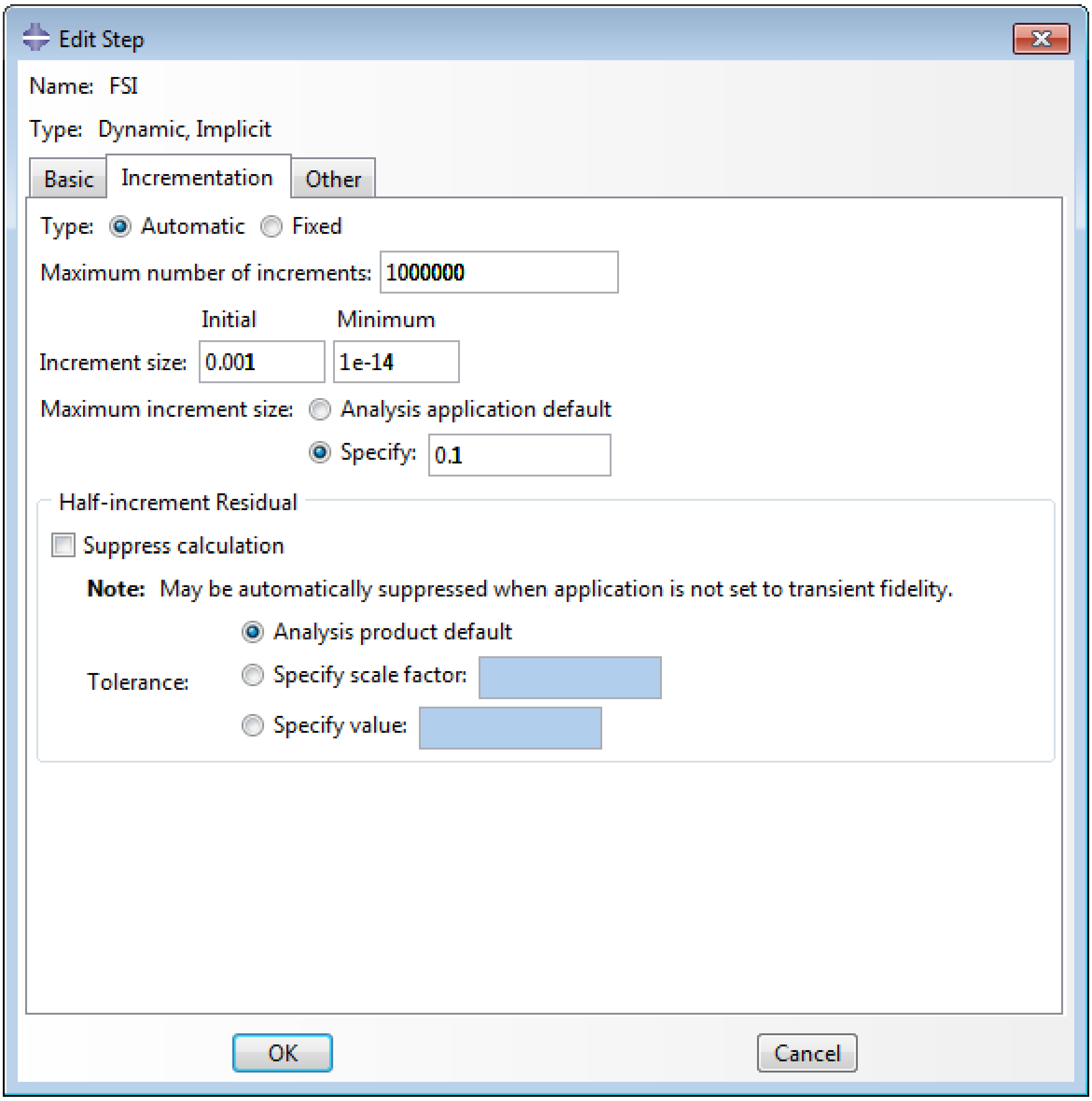
In the **Incrementation** tab specify:

Maximum numbers of increments = 1E6 (maximal number of increments)

Increment size > Initial = 0.001 (initial increment)

Increment size > Minimum = 1E-14 (minimal increment)

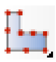
Maximum increment size > Specify = 0.1 (maximal increment)

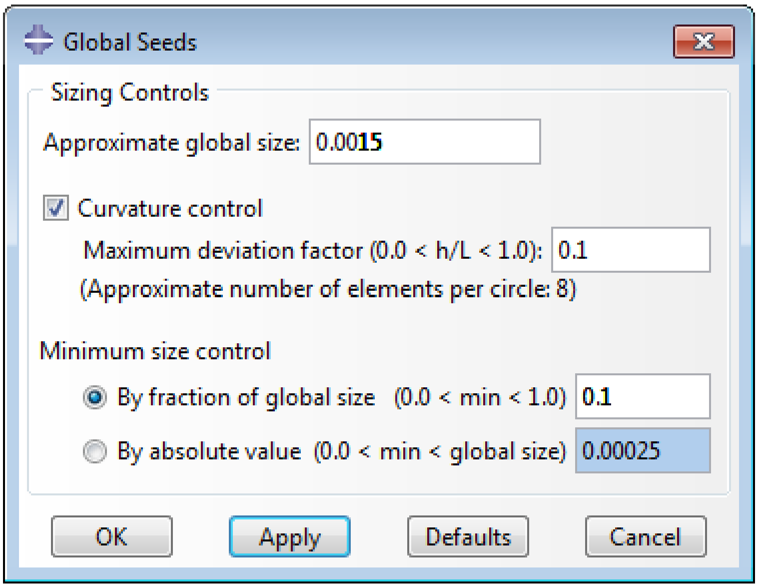


Specifying the procedure of analysis. Creation the FSI step. (3)

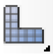
Click **OK**.

11. Navigate to the module **Mesh** and apply the tool  (**Partition Cell: Extrude/Sweep Edges**) to split the valve into two parts. This allows obtaining more structured finite element mesh. Select a boundary that will be used for splitting and the direction, along which the boundary will be moved (**Sweep Along Edge**). Click **Done**.

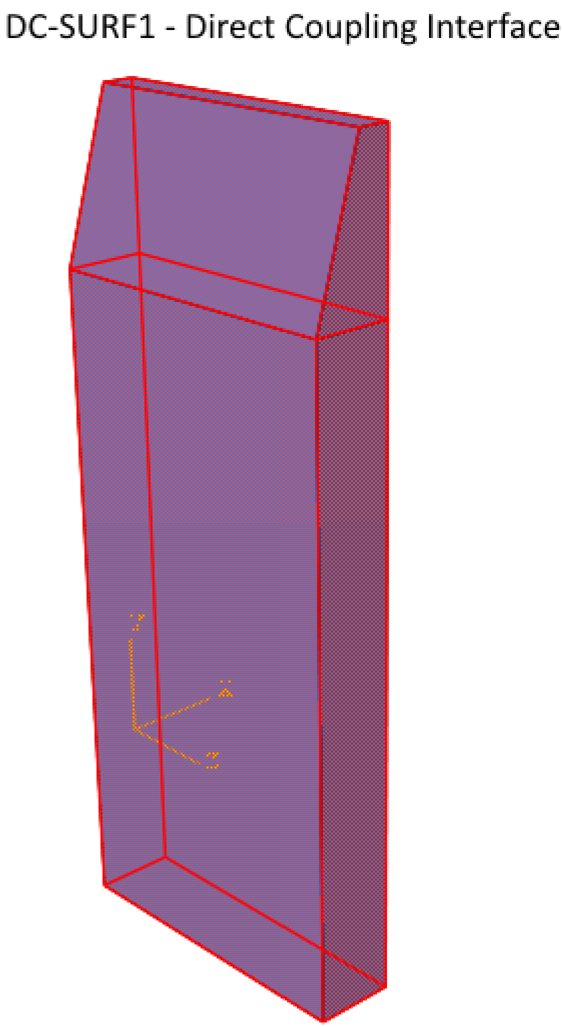
12. Apply the tool  (**Seed part Instance**) to split the valve into elements. The **Global Seeds** dialog box will open. Specify there **Approximate global size = 0.0015** and click **OK**.



Specifying the size of finite elements

13. To start the grid generation, click  (**Mesh Part Instance**) and then click **Yes**.

6.1.1.2 Specifying an interface surface in Abaqus



The next important step of creation the model for joint computation is specifying an ***interface surface*** between *Abaqus* and *FlowVision*.

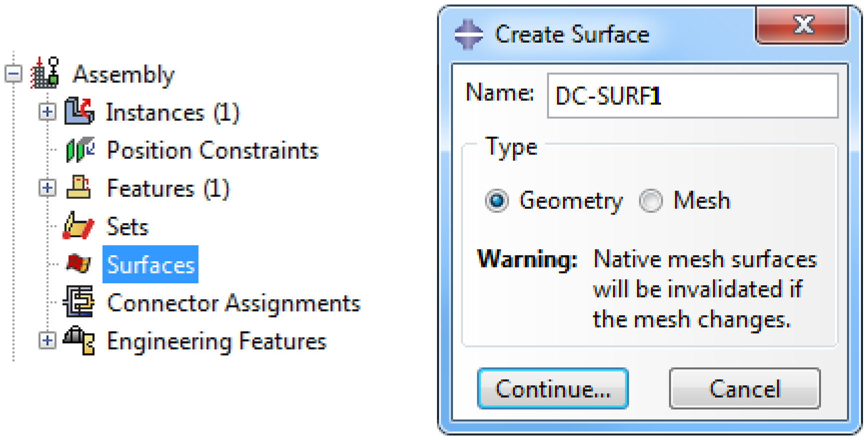
Abaqus simulates deformations of the valve and exports coordinates of nodes of the deformed surface.

Follow these steps:

In the model tree double-click **Assembly > Surfaces** (or select and use the **Create** command). The **Create Surface** dialog box will open; specify there:

Name = DC-SURF1 (name of the interface surface)

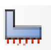
Type = Geometry



Click **Continue** and select the whole valve. Click **Done**.

6.1.1.3 Specifying boundary conditions and loads in Abaqus

Abaqus imports loads (CF) that were calculated in *FlowVision*. Follow these steps:

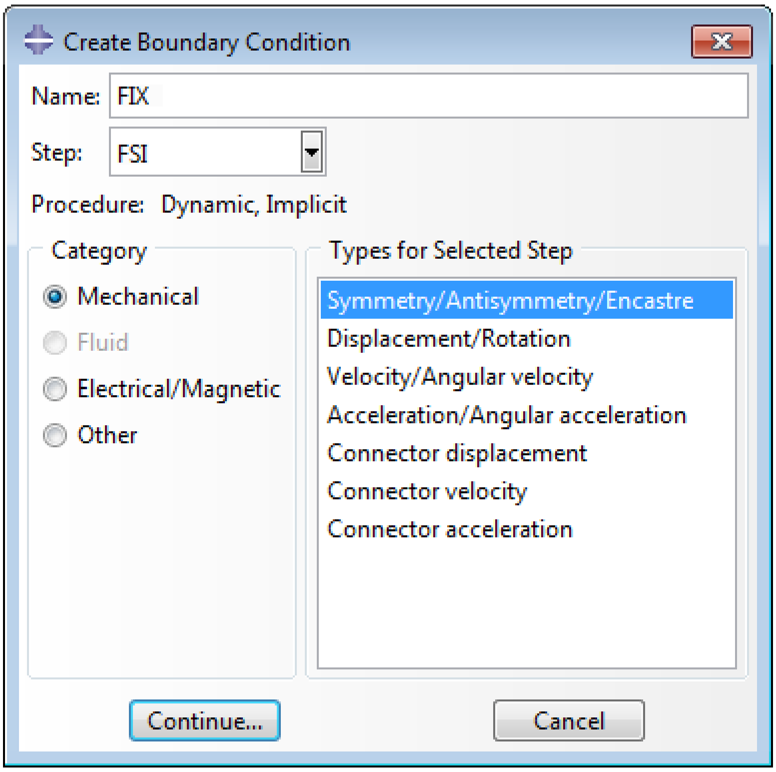
1. Navigate to the module **Load**. Select  (**Create Boundary condition**) and create a boundary condition on the bottom surface of the valve with the following parameters:

Name = FIX

Step = FSI

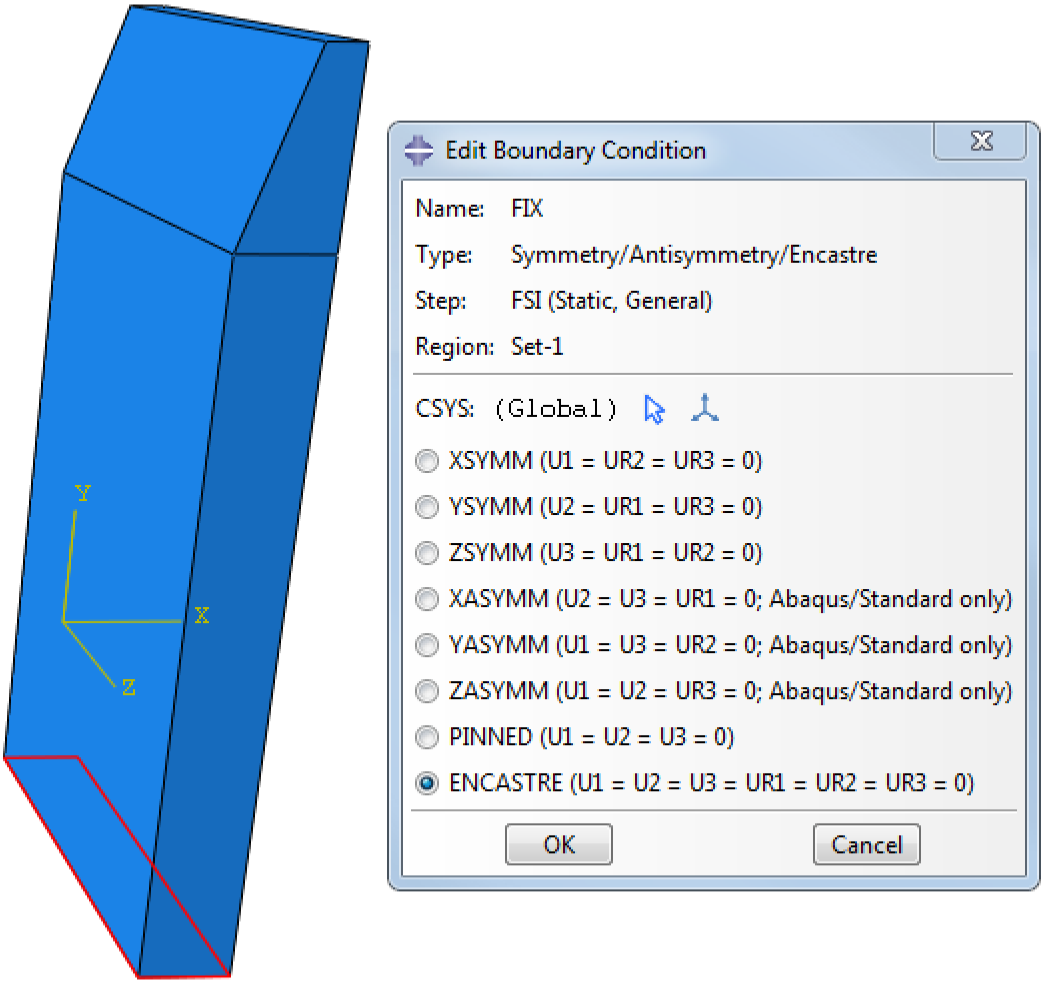
Category = Mechanical

Types for Selected Step = Symmetry/Antisymmetry/Encastre



Click **Continue**.

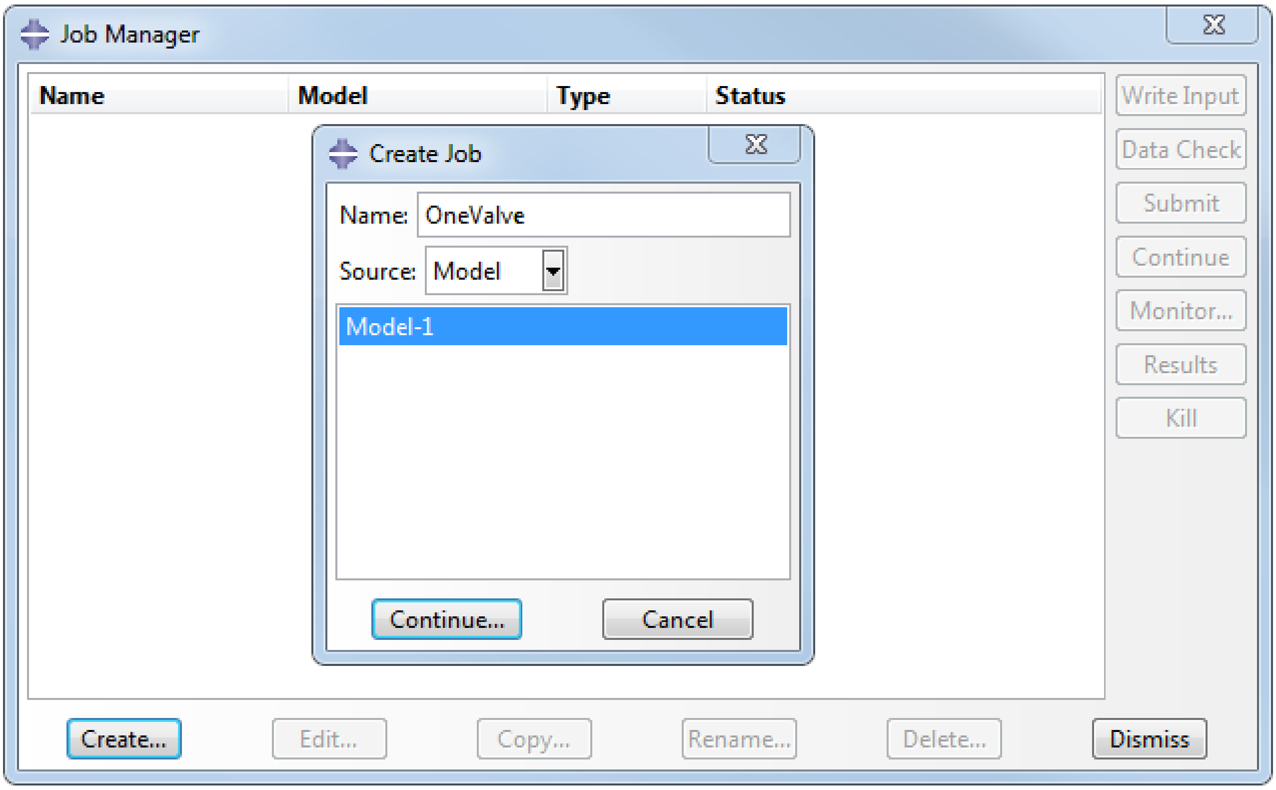
2. In the graphical work window select the bottom surface of the valve and click **Done**. In the **Edit Boundary Condition** dialog box, which opens, select for the valve's face the boundary condition type **ENCASTRE (U1=U2=U3=UR1=UR2=UR3=0)** that prohibits movement of this face for any degrees of freedom:



6.1.1.4 Generating an inp-file

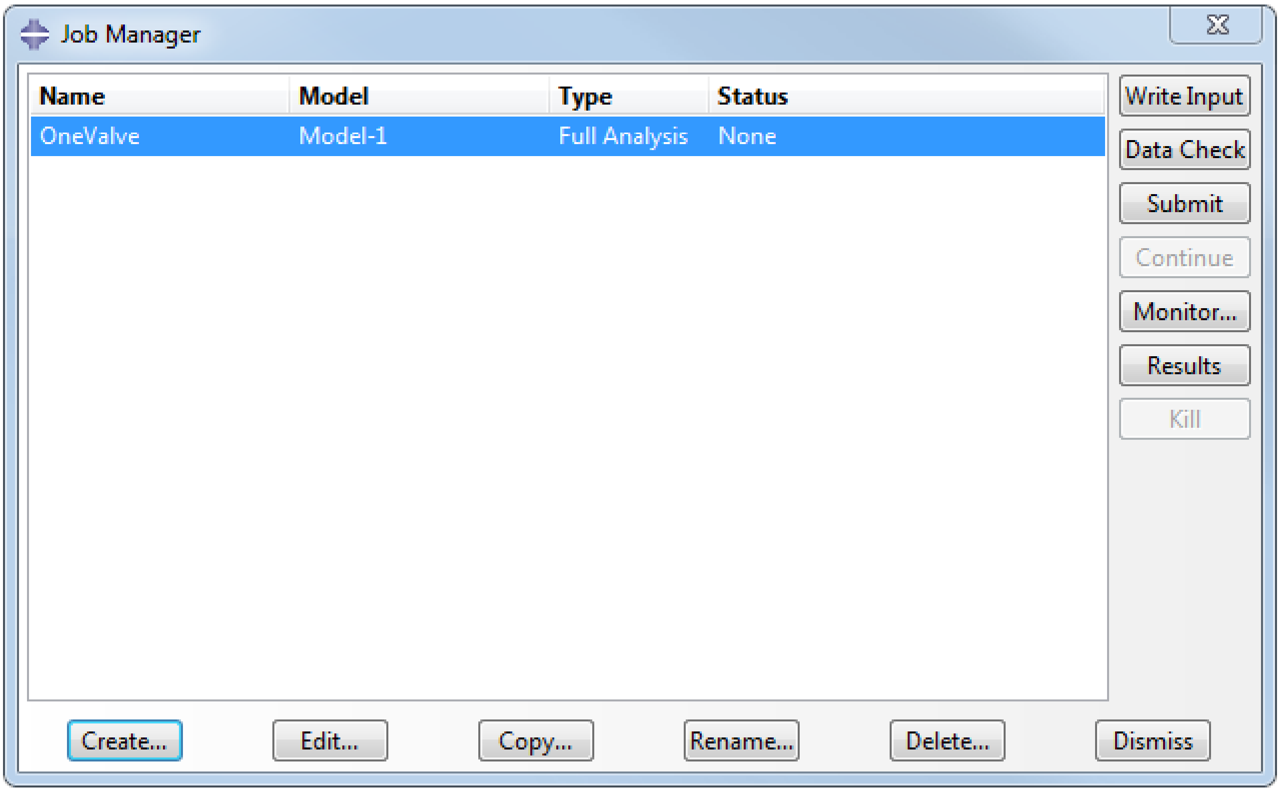
To generate an `inp`-file, follow these steps:

1. Navigate to the module **Job** and in the window **Job Manager** () create a new analysis:



Click **Create**; after this the **Create job** dialog box will open; specify there the name **OneValve** for the new analysis and click **Continue**.

2. In the **Job Manager** window select the line of the just created analysis and click **Write Input** to create an input file, geometry of which will be imported to *FlowVision*. Also this input file will be required to make the computation.

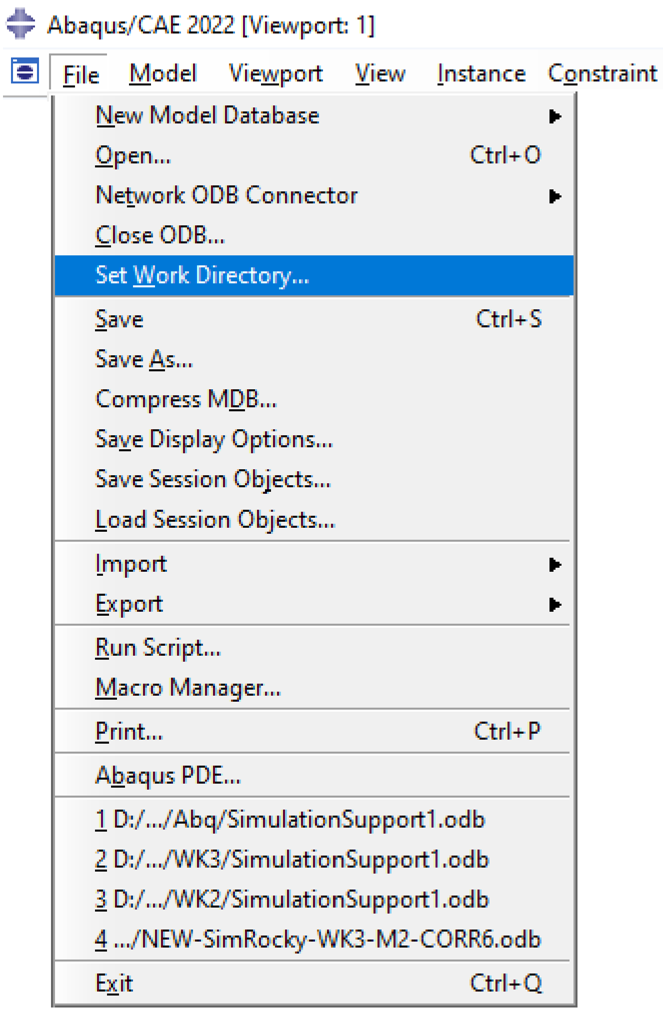


3. Save and close the project.

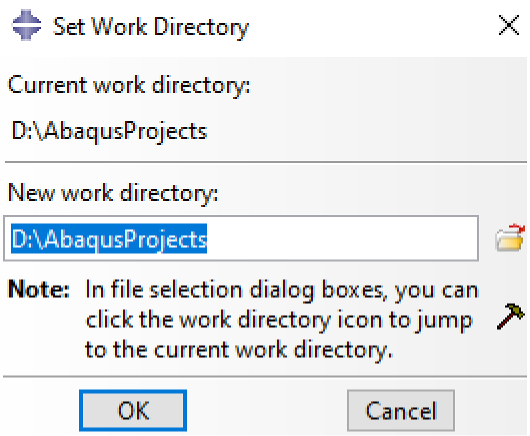
6.1.1.5 Modifying the inp-file of the Abaqus' project

The ready file **`OneValve.inp`**, which is included into the delivery, has already been modified. If you use the file, you can skip this section, but we recommend to open and browse contents of this file using any text editor.

The **`inp`** file, which has been generated, locates in the work directory of *Abaqus*. The work directory of *Abaqus* can be specified by the **File > Set Work Directory** menu command:



You can specify any directory, which is available for writing, as a new work directory of *Abaqus*:



If you wish to create the **`OneValve.inp`** file by your own, follow these steps:

- Open the **inp**-file of the *Abaqus*' project in a text editor.
- Add the following lines into the module **STEP** before the line ***End Step**:

```
**
*CO-SIMULATION CONTROLS, NAME= COSIM_CONTROLS, TIME INCREMENTATION=SUBCYCLE, TIME MARKS=YES
**
*CO-SIMULATION, PROGRAM=DIRECT, NAME=FlowVision, CONTROLS=FSI
*CO-SIMULATION REGION, IMPORT
ASSEMBLY_DC-SURF1, CF
*CO-SIMULATION REGION, EXPORT
ASSEMBLY_DC-SURF1, COORD
*CO-SIMULATION CONTROLS, NAME=FSI, TIME INCREMENTATION=SUBCYCLE, TIME MARKS=YES
**
```

These lines are used to identify the coupling analysis with another program.

Important notice: Add **ASSEMBLY_** before the interface region's name in lines of the co-simulation. After the modification the **inp** file have not to contain empty lines.

See also: More details about these settings (***CO-SIMULATION**, ***CO-SIMULATION CONTROLS**, ***CO-SIMULATION REGION**) you can found in the *Abaqus Keywords Reference Manual.*

6.1.2 Preparing the project in FlowVision

FlowVision simulates motion of the fluid in the channel.

Requirements to the *FlowVision's* project:

- The geometry of the deformable body has to be loaded from *Abaqus* into *FlowVision*, and this geometry must completely comply to the deformable geometry in the *Abaqus* project.
- The geometry of the deformed body, loaded into *FlowVision*, must comply with the requirements to the geometry in *FlowVision* (see *User's guide*).

6.1.2.1 Physical model

In this exercise, the **Substance** is water. The **Substance** will be loaded from the standard **Substance database**.

- Open the context menu of the folder **Substances** and, by the command **Create**, create **Substance #0**.
- From the context menu of **Substance #0** select the command **Load from SD > Standard** and load the substance **Water** in its liquid phase.

In the folder **Phases**:

- Create a continuous **Phase #0**.
- In **Phase #0** add **Substance #0** into the subfolder **Phase #0 > Substances**.
- In properties of the folder **Phase #0 > Physical processes** specify:

Motion	**= Navier-Stokes model**
Turbulence	**= KES**

In the folder **Models**:

- Create **Model #0**.
- Add **Phase #0** into subfolder **Model #0 > Phases**.
- Specify motion of the liquid along the axis X with velocity 10 [m/s]. To do so, in properties of the element **Models > Model #0 > Init. data > Init. data #0 > Velocity (Phase #0)** specify:

X **= 10** [m s^{-1}]

In properties of **SubRegion #0** specify **Model = Model #0**.

6.1.2.2 Imported valve as a Moving body

The valve is inserted into the project as a separate geometry **Object**, on which the **Moving body** modifier is set.

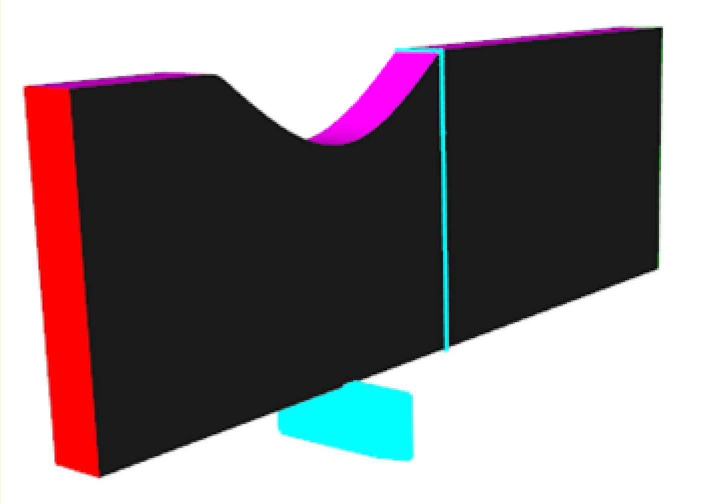
Follow these steps:

In the folder **Objects**:

- From the context menu of the folder **Objects** select the command **Create** and in the **Create new object** dialog box, which opens, select **Object type = Imported object**.
- Import your modified file `OneValve.inp` from your directory or import the provided file `OneValve.inp` from the directory with examples.
- **Imported object #0** will appear in the folder **Objects**.

When an object is imported from *Abaqus*, inconsistencies of the loaded geometry's scale or space orientation can occur relating to other geometry objects from the *FlowVision*'s project.

So you can see that, after inserting the imported valve, the valve requires a shift and rotation to be placed in the flow channel of the tube.



It is important that, in co-simulation, the scaling, shift and rotation of an object must be specified in properties of the **Moving body** modifier, which has been set on an **Imported object**. When you do so, the specified scaling, shift and rotation will be automatically applied to all further replaces of the geometry, received from *Abaqus*.

Create a **Moving body** modifier on **Imported object #0**:

- From the context menu of the folder **Subregions > SubRegion #0 > Modifiers** select the command **Create** and in the **Create new modifier** dialog box, which opens, select:

  Modifier type = Moving body

  Objects = Imported object #0

  The **Moving body #0** modifier will appear in the folder **Subregions > SubRegion #0 > Modifiers**.
- In properties of the just created modifier **Moving body #0** specify its initial position (the initial position of the valve):

Initial position

Reference point

X	**0**	[m]
Y	**-0.001**	[m]
Z	**-0.009**	[m]

Axis Y

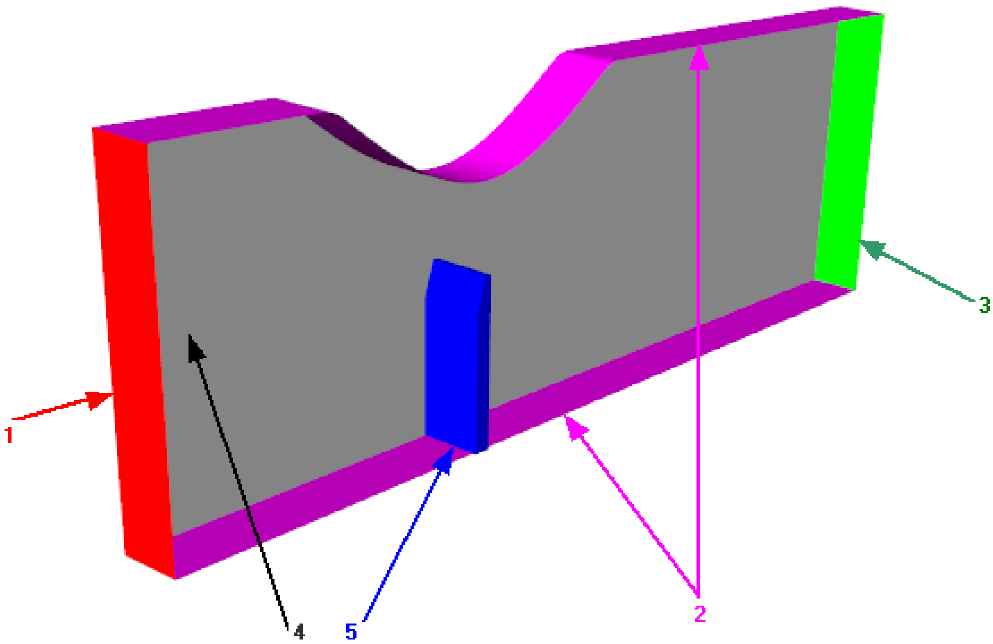
X	**= 0**
Y	**= 0**
Z	**= -1**

- Click **Apply** and then click the icon **Operations >**  (**Place to initial position**).

Note that outer dimensions of the valve are beyond geometry surface of the tube's flow channel. This is requirement to the geometry of *FlowVision* for correct generation of the computational grid: it is not possible to use same surfaces for the **Region** and a **Moving body**, the surfaces must have some mandatory gap between them or some guaranteed overlap.

For correct operation of the joint computation, it is allowed to use only one **Boundary condition** per an exchanged **Moving body**. For this **Boundary condition** you must specify a color, which differs from colors of other **Boundary condition** in the computational domain.

6.1.2.3 Boundary conditions



Specify the following boundary conditions:

Boundary 1

Type	**= Inlet/Outlet**	
Color	**= Red**	
Variables		
Velocity(Phase #0)	**= Normal mass velocity**	
Mass velocity	**= 10000**	[kg m^{-2} s^{-1}]
TurbEnergy(Phase #0)	**= Pulsations**	
Value	**= 0**	
TurbDissipation(Phase #0)	**= Turbulent scale**	
Value	**= 0**	[m]

Boundary 2

Type	**= Wall**
Color	**= Fuchsia**
Variables	

Velocity(Phase #0) **= Logarithm law**
TurbEnergy(Phase #0) **= Value in cell near wall**
TurbDissipation(Phase #0) **= Value in cell near wall**

Boundary 3
Type **= Free Outlet**
Color **= Green**
Variables
Velocity(Phase #0) **= Pressure**
Value **= 0** [Pa]
TurbEnergy(Phase #0) = ***Zero gradient***
TurbDissipation(Phase #0) = ***Zero gradient***

Boundary 4
Type **= Symmetry**
Color **= Gray**
Variables
Velocity(Phase #0) **= Slip**
TurbEnergy(Phase #0) **= Symmetry**
TurbDissipation(Phase #0) **= Symmetry**

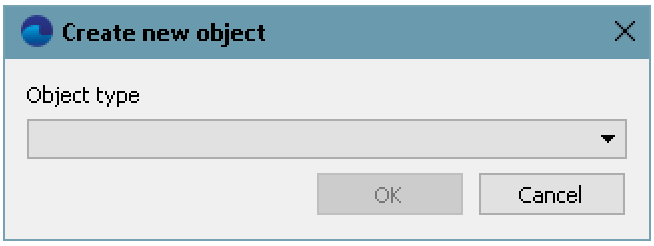
Boundary 5
Name **= Valve**
Type **= Wall**
Color **= Blue**
Variables
Velocity(Phase #0) **= Logarithm law**
TurbEnergy(Phase #0) **= Value in cell near wall**
TurbDissipation(Phase #0) **= Value in cell near wall**

Important note: one single boundary condition is set to carry out the joint computation on the whole surface of the replaceable geometry object (the valve).

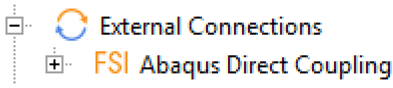
6.1.2.4 Parameters of co-simulation

- From the context menu of the **External Connections** folder select the **Create** command:

- In the **Create new object** dialog box, which opens, select **Object type = Abaqus Direct Coupling**:



* An operating system's dialog box for access to files will open; select there the `OneValve.inp` file, which was crested on the step Generating an inp-file.
* An **Abaqus Direct Coupling** child element will appear in the **External Connections** folder:

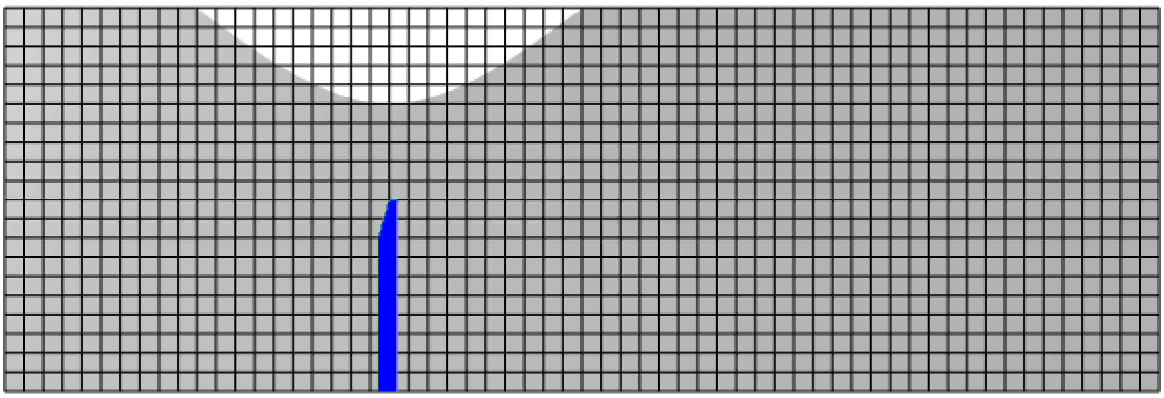


The **Abaqus Direct Coupling** element, in its turn, has a child element **ASSEMBLY_DC-SURF1**.

* In properties of the folder **External Connections** specify:
  Exchange step > Method = User value
  Exchange step > Constant step = 0.01
  Exchange step > Coef. for time step = 1
  Old offset = Yes
  AutoSave = Yes
* In properties of the element **Abaqus Direct Coupling** specify:
  Activation = Yes
  ABAQUS > Run ABAQUS = Yes
  ABAQUS > MPM Agent > Address = Host name or IP address of the computer, on which *MPM Agent* will run. *MPM Agent* will start *Abaqus*.
  ABAQUS > MPM Agent > Port = value of this field is filled automatically
  ABAQUS > ABAQUS-project = network path and name of the file with the *Abaqus* project (when it runs on a remote computer, specify the path to the project on the remote computer)
  ABAQUS > IP Source = IP of MPM Agent
  ABAQUS > Port = 5555
  Loads relaxation > Scale factor = 1
  Loads relaxation > Start in steps = 0
  Loads relaxation > End in steps = 10
  Loads relaxation > Initial coefficient = 0
  Loads relaxation > Final coefficient = 1
  Heat relaxation > Scale factor = 1
  Heat relaxation > Start in steps = 0
  Heat relaxation > End in steps = 0
  Heat relaxation > Initial coefficient = 1
  Heat relaxation > Final coefficient = 1
* In properties of the element **ASSEMBLY_DC-SURF1** specify **Moving body = Moving body #0**.

Don't change other settings.

6.1.2.5 Initial grid



In properties of the **Initial grid** specify:

Grid structure = 2D

Plane = XZ

nX = 60

nZ = 20

In the **Properties** window of the **Initial grid** click **Apply**.

6.1.2.6 Adaptation

In this simulation, it is necessary to split grid cells near surface of the valve and merge the previously adapted cells, which locate away from the moving valve.

To refine the computational grid near the valve, an adaptation over the surface of an imported object is used. Specify this adaptation as it is described in the following steps:

- Create the element **Computational grid > Adaptation > Adaptation #0**.
- Add **Imported object #0** into the folder **Computational grid > Adaptation > Adaptation #0 > Objects**.
- In properties of **Adaptation #0** specify:

Enabled	**= Yes**
Max level N	**= 2**
Layers	
Layers for Level N	**= 4**
Layers for Level N-1	**= 4**

When the valve moves, the area around it, where adaptation is being done, also moves. To avoid useless adaptation away from the valve, you should specify merging of cells there. During the merging the previously adapted cells are restored to their initial level of adaptation. Thus only those cells remain adapted, which locate near the valve.

Specify merging of the previously split cells:

- Create the element **Computational grid > Adaptation > Adaptation #1**.
- Add the geometry object **Computational space** to the folder **Computational grid > Adaptation > Adaptation #1 > Objects**.
- In properties of **Adaptation #1** specify:

Enabled	**= Yes**
Max level N	**= 0**
Split/Merge	**= Merge**

6.1.2.7 Parameters of calculation in FlowVision

For simulations that include motion of moving bodies (FSI computations are such ones), we recommend to specify the time step by the CFL number (specify the surface CFL as 1 and convective CFL in the range from 1 to 100 depending on the problem).

In the **Solver** tab, in properties of the **Time step** element, specify:

Method	**= Via CFL number**
Convective CFL	**= 100**
Surface CFL	**= 1**

6.1.2.8 Visualization

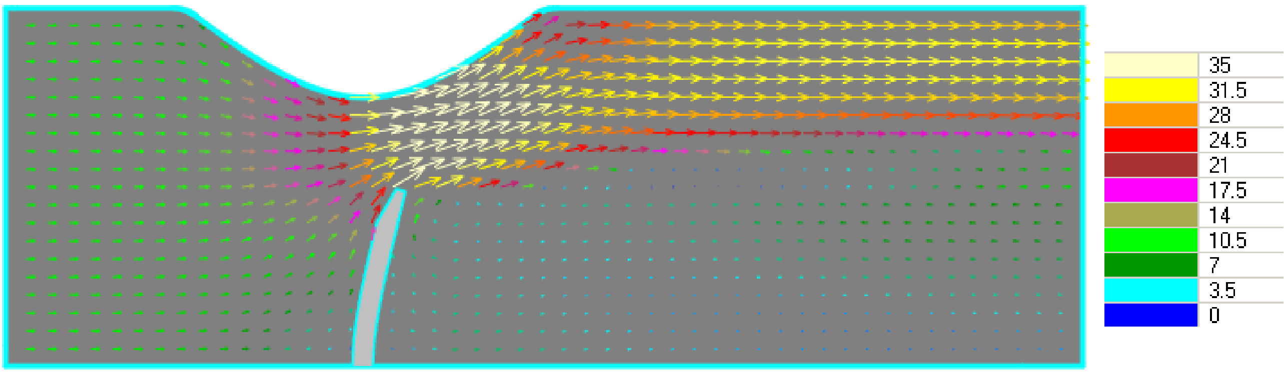
Viewing the results of the computation is possible both during the computation (using the **Viewer** module of *FlowVision*) and after finishing the computation.

To view the dynamics of the solution during the computation, prior to the computation, specify visualizations of:

1. Velocity distribution in the plane of the flow.
2. Pressure distribution on the surface of the valve

In its current version **Pre-Postprocessor** can not visualize deformed geometry during the computation (this functionality is under construction). Visualization of the deforming geometry during the computation can be done using the *FlowVision*'s **Viewer** module. When **Pre-Postprocessor** works with a completed computation, the deforming geometry *is* visualized.

6.1.2.8.1 Velocity distribution



Vectors of velocities in the plane of the flow

- In the properties of **Plane #0**, specify:

Object

Normal

X	**= 0**
Y	**= 1**
Z	**= 0**
Shift	**= 0.01**

- Create the layer **Vectors #0** on **Plane #0**.
- In the properties of the layer **Vectors #0**, specify*):

Grid

Size 1	**= 50**
Size 2	**= 20**

Coloring

Variable

Variable	**= Velocity**

Value range

Mode	**= Manual**
Max	**= 35**
Min	**= 0**

Palette

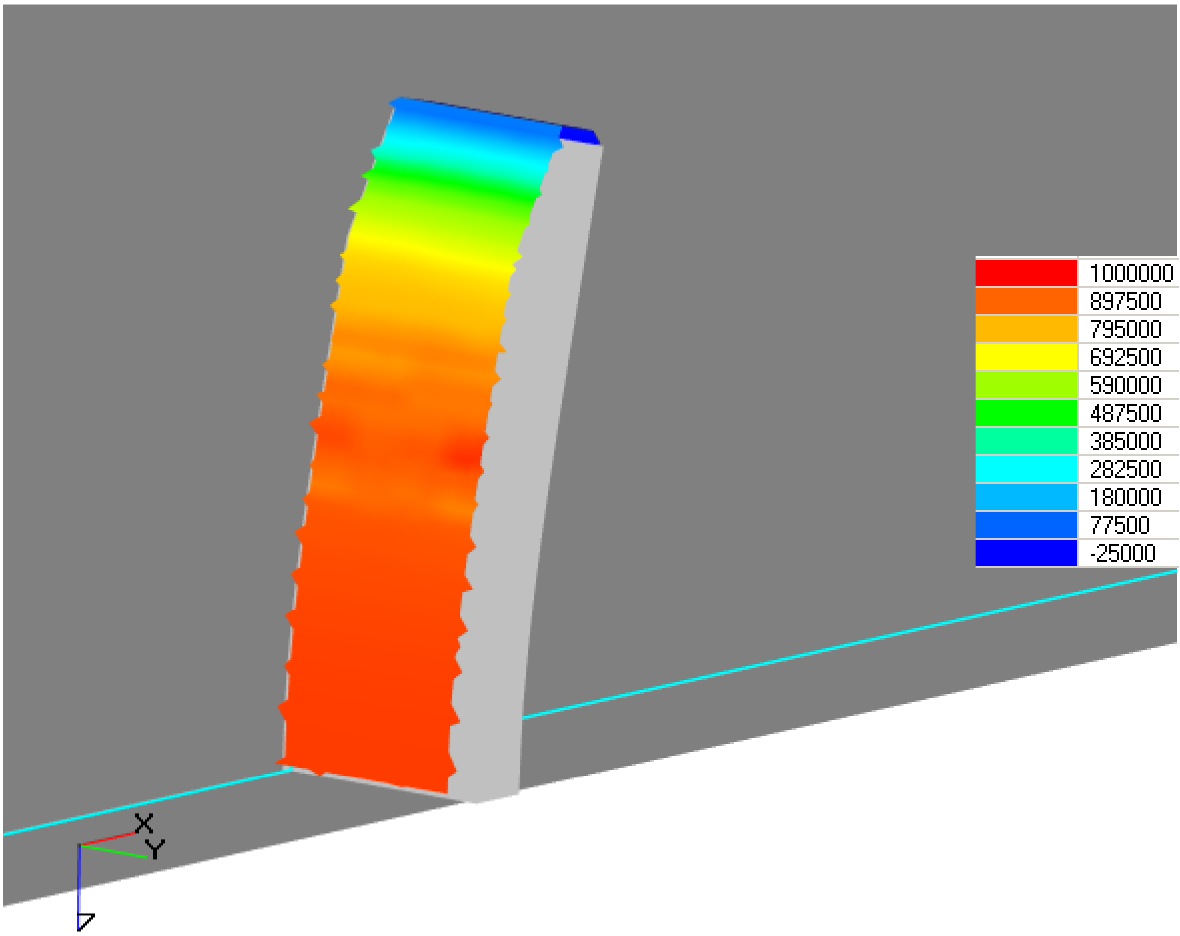
Operations	Click (**Load palette from file**) and then select the file `heat.fvpal` (this file locates in the directory where *FlowVision* is installed).

*)The program will automatically specify the variable, which is used to build the vectors: **Variable > Variable = Velocity**.

Notes:

- To display the palette, which is used for coloring the layer (in our case this is coloring the vectors by their absolute values), you can use the **Info** window or, in properties of the layer, set **Appearance > Enabled = Yes**. To open the **Info** window, you have to select the layer in the tree of **Postprocessor** and click the button.
- If you wish to hide some layers or objects displayed in the **View** window, use the **Hide** command from their context menus.

6.1.2.8.2 Pressure distribution



Distribution of pressure on the surface of the valve

- Create a **Color contours** layer on **Imported object #0**.
- In the properties of the just created layer **Color contours #0**, specify:

Variable	
Variable	**= Pressure**
Value range	
Mode	**= Manual**
Max	**= 1000000**
Min	**= -25000**

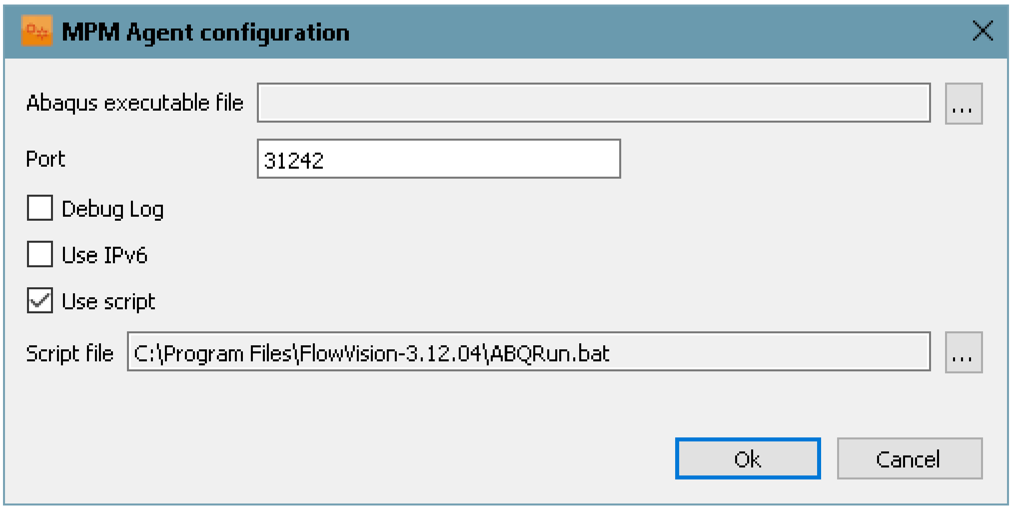
Notes:

- To display the palette, which is used for coloring the layer, you can use the **Info** window or, in properties of the layer, set **Appearance > Enabled = Yes**. To open the **Info** window, you have to select the layer in the tree of **Postprocessor** and click the button.
- If you wish to hide some layers or objects displayed in the **View** window, use the **Hide** command from their context menus.

6.1.3 Starting and stopping the computation

Before starting the computation you have to run **MPM-Agent**.

MPM-Agent has to receive the path to the executable file of *Abaqus*. If the path has not been provided, you can specify it in the *FlowVision*'s module **Configurator**. To do so, click in **Configurator** the **View** button on the right near the **MPM Agent** field in the **Configuration files** pane in the **Configuration/Logs** tab and specify the data in the **MPM Agent configuration** dialog box, which opens:



The project's computation is started from **Pre-Postprocessor**, see description of a standard starting in the section **Detailed description of a simplest model > Laminar flow in a tube > Starting the computation**.

Stopping after 20 seconds of the simulated time is set in Abaqus. Also you can stop the project using the (**Stop computation**) button, and then resume the computation after editing the project.

6.2 Two valves in a channel

In this exercise we examine *FlowVision-Abaqus* co-simulation of motion of two deformable valves, which obstruct the flow.

For this the computation uses two **Moving bodies**.

Problem setting

Two valves, which are placed into the channel, obstruct the flow of the liquid. The joint computation made by *Abaqus* and *FlowVision* simulates motion of the valves and parameters of the flow in mutual interaction.

This exercise it a modification of the previous exercise (Deformable valve in channel), to which another **Imported object** is added representing the second valve.

Files:

Geometry model of the tube	`Valve_Channel.wrl`
Geometry of valves	`Valve1.stl, Valve2.stl`
Project in *Abaqus*	`TwoValves.inp`
Project in *FlowVision*	`Two_Valves_Channel`

6.2.1 Preparing the project in Abaqus

Follow the steps described in the sections below:

- Creating the Abaqus project
- Export of geometries
- Modifying the inp-file of the Abaqus' project

6.2.1.1 Creating the Abaqus project

Begin creation of the *Abaqus* project for two valves with repeating the procedure, which has been described in the "Deformable valve in channel" exercise.

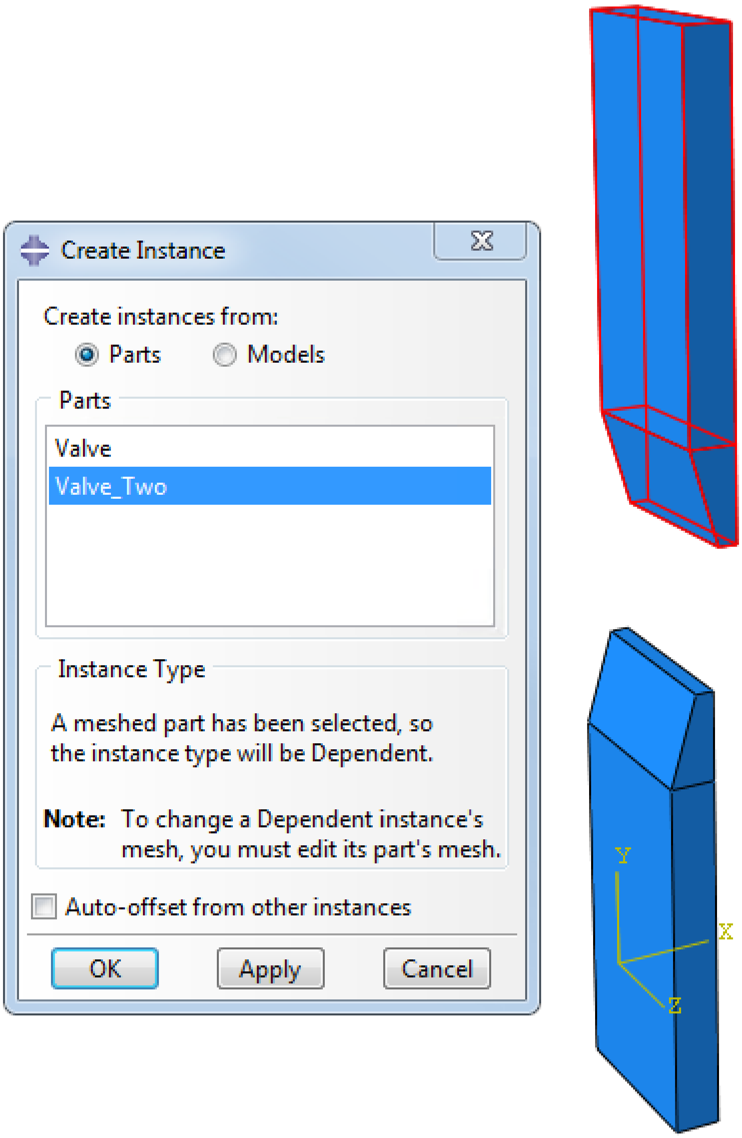
Then, in the **Part** module, create a second part with following coordinates:
(-0.003,0.13);
(0.002,0.13);
(0.002,0.08);
(0,0.08);
(-0.003,0.09)

and with width **20e-3**.

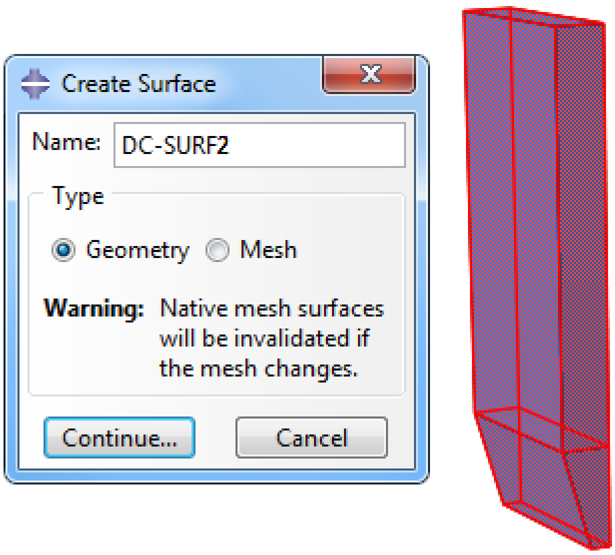
In the **Property** module, assign to this part the section, which was created before in the exercise with one valve.

In the **Mesh** module, create a finite element mesh for this part similarly as it was done for the first valve.

In the **Assembly** module, add the second valve into the assembly.



Create a separate close surface for the second valve. Name this surface as **DC-SURF2**.



Assign to this valve the same boundary condition as those, which has been assigned to the first valve.

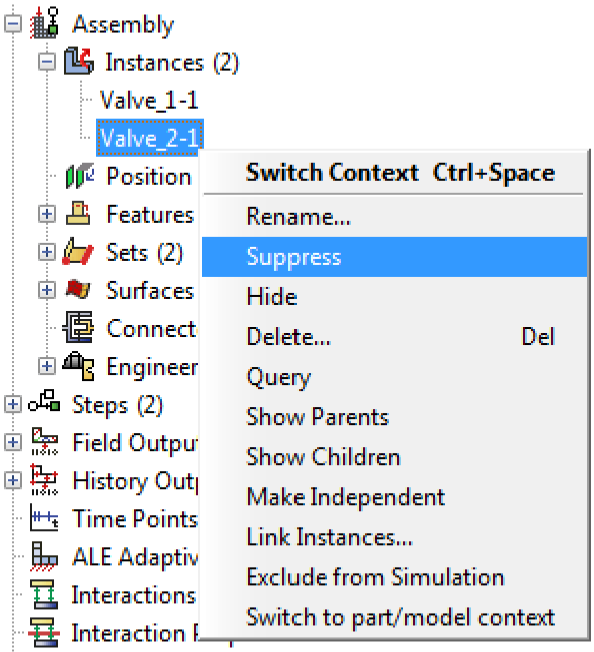
6.2.1.2 Export of geometries

In co-simulations, *FlowVision* can use any supported file formats that for finite element meshes: **`stl`**, **`dat`**, **`nas`**, **`cdb`**, **`cel`**, **`inp`**, **`mesh`**, **`ngeom`**, **`vtk`**, **`wrl`**.

Because the ready **`inp`**-file already contains geometries of both valves, you have to do a separate import for each of the valves. Otherwise *FlowVision* would import these valves as a single moving body.

One of the methods for exporting the geometry is creation **`stl`** files.

In the **Assembly** module, in the context menu of the second valve, select **Suppress**. The second valve will be hidden in the assembly.



Select the **Plug-ins > Tools > STL Export** menu item and export the first valve with name **`Valve1.stl`**.

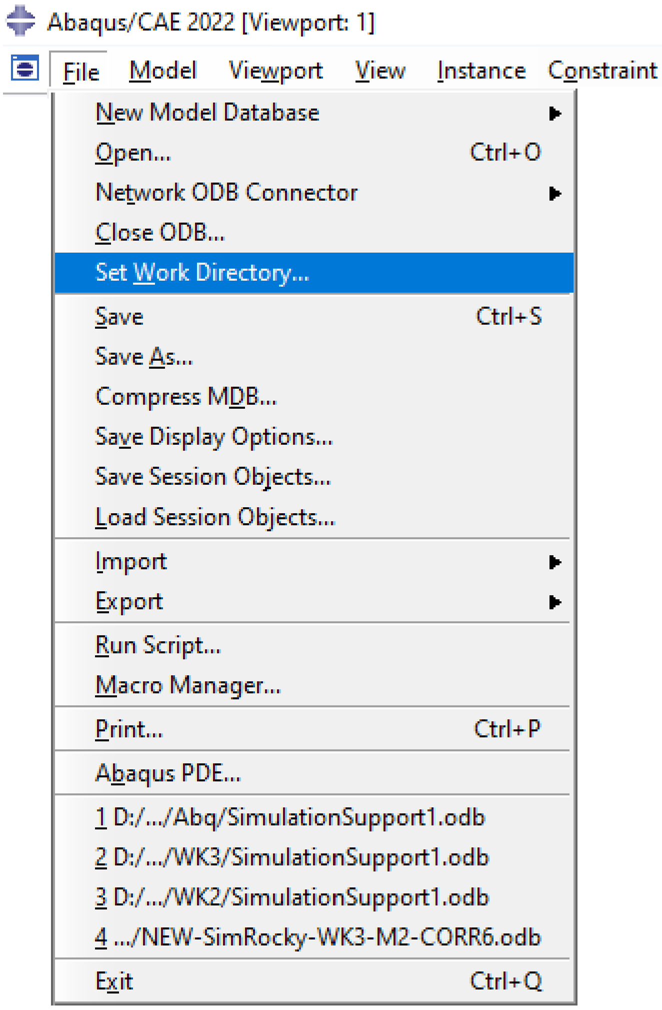
Then display the second valve (select **Resume** from its context menu) and hide the first valve (select **Suppress** from its context menu). Export the second valve with name **`Valve2.stl`**.

After exporting the geometries, display the first valve and generate the file **`TwoValves.inp`**.

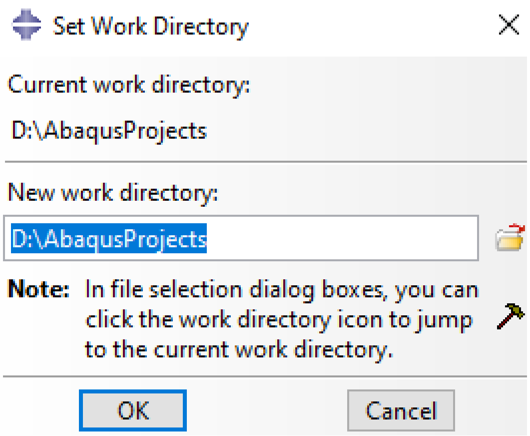
6.2.1.3 Modifying the inp-file of the Abaqus' project

The ready file **`TwoValves.inp`**, which is included into the delivery, has already been modified. If you use the file, you can skip this section, but we recommend to open and browse contents of this file using any text editor.

The **`inp`** file, which has been generated, locates in the work directory of *Abaqus*. The work directory of *Abaqus* can be specified by the **File > Set Work Directory** menu command:



You can specify any directory, which is available for writing, as a new work directory of *Abaqus*:



If you wish to create the **`TwoValves.inp`** file by your own, follow these steps:

- Open the **`inp`**-file of the *Abaqus*' project in a text editor.
- Add the following lines into the module **STEP** before the line **`*End Step`**:

```
*CO-SIMULATION, PROGRAM=MULTIPHYSICS, Name=FlowVision
*CO-SIMULATION REGION, IMPORT
ASSEMBLY_DC-SURF1, PRESS
ASSEMBLY_DC-SURF2, PRESS
*CO-SIMULATION REGION, EXPORT
ASSEMBLY_DC-SURF1, U
ASSEMBLY_DC-SURF2, U
```

These lines are used to identify the coupling analysis with another program.

Note that two interface surfaces (**DC-SURF1** and **DC-SURF2**) are simulated in this computation. Save and close the project.

Important notice: Add `ASSEMBLY_` before the interface region's name.

See also: More details about these settings (`*CO-SIMULATION`, `*CO-SIMULATION REGION`) you can found in the *Abaqus Keywords Reference Manual.*

6.2.2 Preparing the project in FlowVision

Follow the steps described in the sections below:

- Physical model
- Imported objects
- Boundary conditions
- Settings for co-simulation
- Generating the CSE Director's configuration file
- Computational grid and its adaptation
- Parameters of FlowVision's calculation

6.2.2.1 Physical model

In this exercise, the **Substance** is water. The **Substance** will be loaded from the standard **Substance database**.

- Open the context menu of the folder **Substances** and, by the command **Create**, create **Substance #0**.
- From the context menu of **Substance #0** select the command **Load from SD > Standard** and load the substance **Water** in its liquid phase.

In the folder **Phases**:

- Create a continuous **Phase #0**.
- In **Phase #0** add **Substance #0** into the subfolder **Phase #0 > Substances**.
- In properties of the folder **Phase #0 > Physical processes** specify:

Motion = **Navier-Stokes model**

Turbulence = **KES**

In the folder **Models**:

- Create **Model #0**.
- Add **Phase #0** into subfolder **Model #0 > Phases**.
- Specify motion of the liquid along the axis X with velocity 10 m/s. To do so, in properties of the element **Models > Model #0 > Init. data > Init. data #0 > Velocity (Phase #0)** specify:

X = **10** [m s^{-1}]

In properties of **SubRegion #0** specify:

Model = Model #0

6.2.2.2 Imported objects

The valves are included into the project as separate geometry **Objects**, on which **Moving body** modifiers are set.

Follow these steps:

In the folder **Objects**:

- From the context menu of the **Objects** folder select the **Create** command and in the **Create new object** dialog box, which opens, select **Object type = Imported object**.
- In the dialog box, which opens, select the file `Valve1.stl`.
- **Imported object #0** will appear in the **Objects** folder.
- Similarly import the second valve from the file `Valve2.stl`.
- **Imported object #1** will appear in the **Objects** folder.

Create **Moving body** modifiers on the **Imported objects**:

- From the context menu of the folder **Subregions > SubRegion #0 > Modifiers** select the **Create** command and in the **Create new modifier** dialog box, which opens, select:

  Modifier type = Moving body

  Objects = Imported object #0

- Similarly create another **Moving body** on **Imported object #1**.
- In properties of **Moving body #0** specify:

  Initial position > Reference point > X = 0

  Initial position > Reference point > Y = -0.001

  Initial position > Reference point > Z = -0.009

  Initial position > Axis Y > X = 0

  Initial position > Axis Y > Y = 0

  Initial position > Axis Y > Z = -1

- Click **Apply** and then click the **Operations >**  (**Place to initial position**) button.
- In properties of **Moving body #1** specify:

  Initial position > Reference point > X = 0.07

  Initial position > Reference point > Y = -0.001

  Initial position > Reference point > Z = 0.029

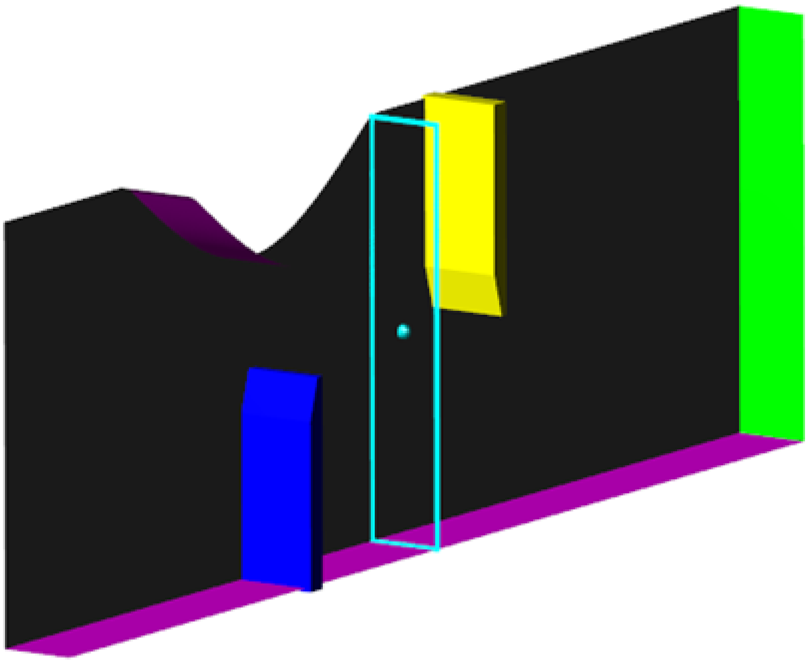
  Initial position > Axis Y > X = 0

  Initial position > Axis Y > Y = 0

  Initial position > Axis Y > Z = -1

- Click **Apply** and then click the **Operations >**  (**Place to initial position**) button.

6.2.2.3 Boundary conditions

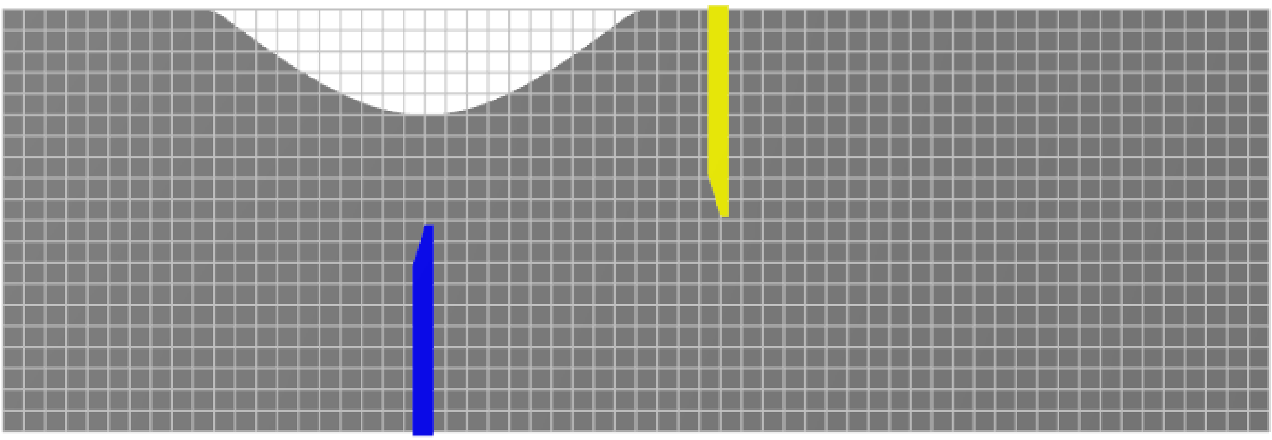


For the tube and the first valve use the same Boundary conditions as there were in the previous exercise (Deformable valve in channel).

Create a new **Boundary condition** for the second valve:

- copy the **Boundary condition** of the first valve
- specify another **Color** for it (for example, yellow)
- assign the new **Boundary condition** to the second valve

6.2.2.4 Computational grid and its adaptation



In the **Properties** window of the **Initial grid** specify:

Grid structure = 2D

Plane = XZ

nX = 60

nZ = 20

In the **Properties** window of the **Initial grid** click **Apply**.

In this simulation, it is necessary to split grid cells near surfaces of the valves and merge the previously adapted cells, which locate away from the moving valves.

To refine the computational grid near the valves, an adaptation over the surface of imported objects is used. Specify this adaptation as it is described in the following steps:

- Create the element **Computational grid > Adaptation > Adaptation #0**.
- Add **Imported object #0** and **Imported object #1** into the folder **Computational grid > Adaptation > Adaptation #0 > Objects**.
- In properties of **Adaptation #0** specify:

Enabled = **Yes**
Max level N = **2**
Layers
  Layers for Level N = **4**
  Layers for Level N-1 = **4**

When the valves move, areas around them, where adaptation is being done, also move. To avoid useless adaptation away from the valves, you should specify merging of cells there. During the merging the previously adapted cells are restored to their initial level of adaptation. Thus only those cells remain adapted, which locate near the valves.

Specify merging of the previously split cells:

- Create the element **Computational grid > Adaptation > Adaptation #1**.
- Add the **Computational space** geometry object into the folder **Computational grid > Adaptation > Adaptation #1 > Objects**.
- In properties of **Adaptation #1** specify:

Enabled = **Yes**
Max level N = **0**
Split/Merge = **Merge**

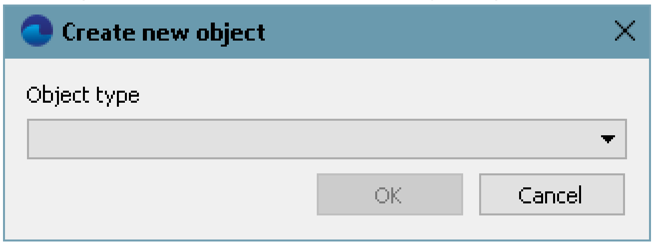
6.2.2.5 Parameters of co-simulation

In this exercise a communication connector (*co-simulation engine*) is used. This connector operates with independent software components, its name is *CSE Director*.

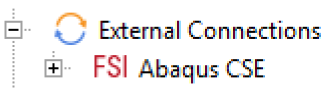
CSE Director defines all aspects of inter-operation between programs and follow the necessary instructions to implement schemes of joint computations.

- In the context menu of the **External Connections** folder select the **Create** command:

- In the **Create new object** dialog box, which opens, select **Object type = Abaqus CSE**:



- The operating system's dialog box for access to files will open; select there your own file `TwoValves.inp`, which you have created created before, or ready `TwoValves.inp` file from the directory with examples. (see section Preparing the project in Abaqus).
- An **Abaqus CSE** child element will appear in the **External Connections** folder:



The **Abaqus CSE** element, in its turn, has two child elements, **ASSEMBLY_DC-SURF1** and **ASSEMBLY_DC-SURF2**.

- In properties of the folder **External Connections** specify:
  Exchange step > Method = User value

Exchange step > Default time step = 0.01
Exchange step > Coef. for time step = 1
Old offset = Yes

- In properties of the element **Abaqus CSE** specify:
  Activation = Yes
  ABAQUS > Run ABAQUS = No
  ABAQUS > IP Source = User
  ABAQUS > Address = Host name or IP address of the computer, on which *Abaqus* will run
  ABAQUS > Port = 5555
  Loads relaxation > Scale factor = 1
  Loads relaxation > Start in steps = 0
  Loads relaxation > End in steps = 10
  Loads relaxation > Initial coefficient = 0
  Loads relaxation > Final coefficient = 1
  Heat relaxation > Scale factor = 1
  Heat relaxation > Start in steps = 0
  Heat relaxation > End in steps = 0
  Heat relaxation > Initial coefficient = 1
  Heat relaxation > Final coefficient = 1
- In properties of the element **ASSEMBLY_DC-SURF1** specify **Moving body = Moving body #0**.
- In properties of the element **ASSEMBLY_DC-SURF2** specify **Moving body = Moving body #1**.

Don't change other settings.

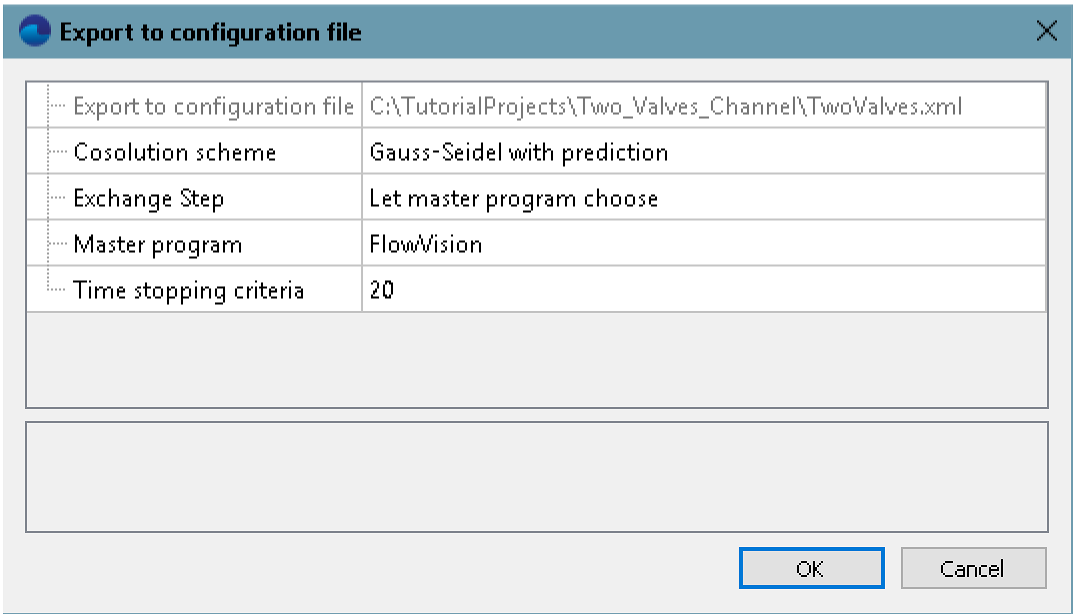
6.2.2.6 Generating the CSE Director's configuration file

To connect the programs, *CSE Director* uses a configuration `xml` file, which stores parameters of the joint computation, transferred data and connection between regions.

In *FlowVision* you can generate this file according to your settings. To do so, in properties of the **External Connections > Abaqus CSE** element, click the **Operations >** **(Save configuration file for CS-service)** button.

The **Export to configuration file** dialog box will open; specify there parameters:

- **Cosolution scheme = Gauss-Seidel with prediction**
- **Exchange Step = Let master program choose**
- **Master program = FlowVision**
- **Time stopping criteria = 20**



Click **OK**. In the directory, which contains the client part of the project, the **`TwoValves.xml`** file will appear.

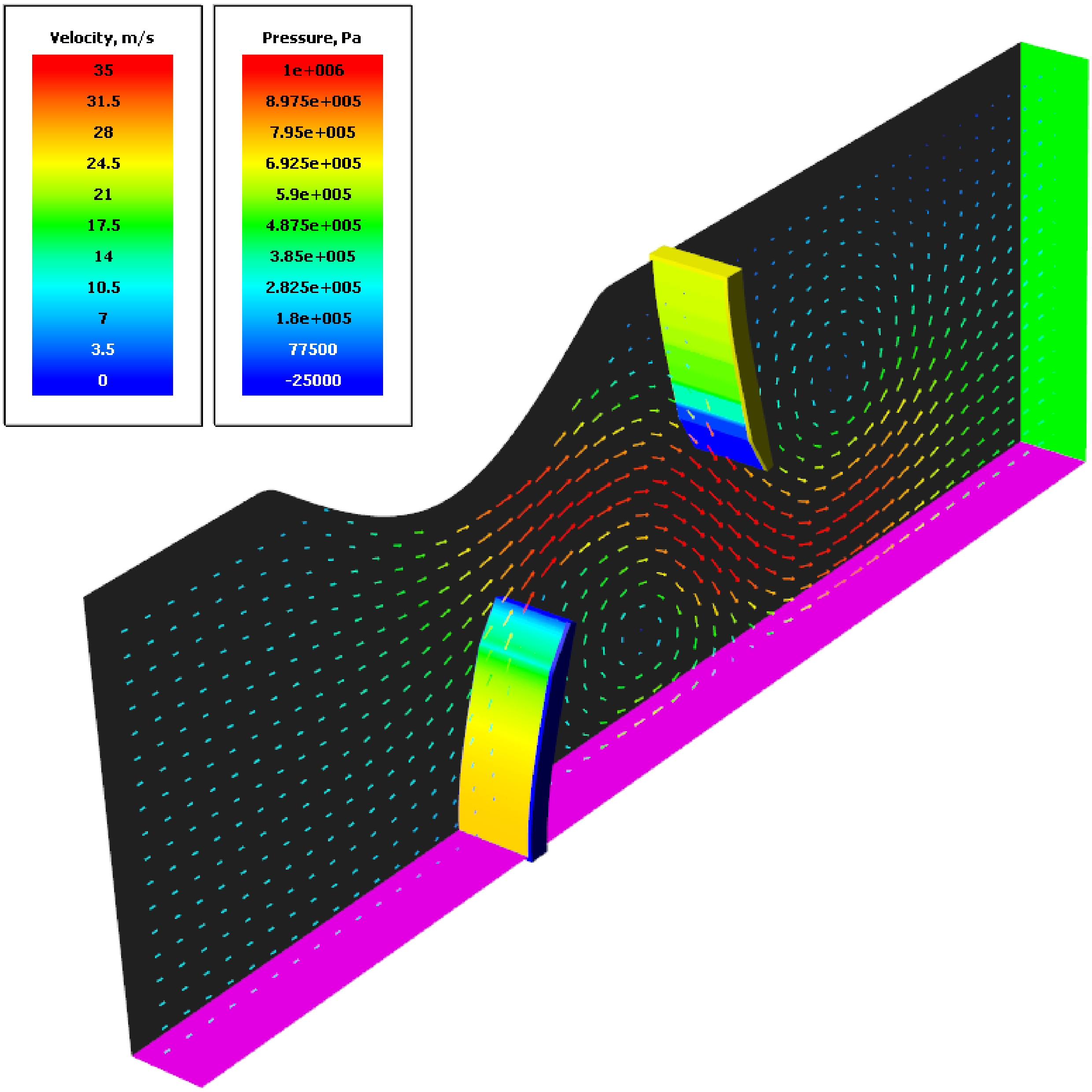
6.2.2.7 Parameters of FlowVision's calculation

For simulations that include motion of moving bodies and, particularly, for FSI computations, we recommend to specify the time step by the CFL number (specify the surface CFL as 1 and the convective CFL in the range from 1 to 100 depending on specifics of the problem).

In the **Solver** tab, in properties of the **Time step** element, specify:

Method	**= Via CFL number**
Convective CFL	**= 100**
Surface CFL	**= 1**

6.2.2.8 Visualization



Apply your skills, formed in the previous exercise (Deformable valve in channel, see the sections Velocity distribution and Pressure distribution), to specify visualization of:

- vectors of **Velocity** of the flowing liquid
- pressure on **Imported objects**

6.2.3 Start of joint computation

- Manual start of Abaqus
- Starting the computation from FlowVision

For this exercise it is possible to start a joint computation using **MPM-Agent** (in *FlowVision 3.13.01* or newer versions). In older *FlowVision* versions only manual start of a joint computation is possible.

To run this exercise using **MPM-Agent**, apply settings described in the exercise Deformable valve in channel (see details in the section Starting and stopping the computation).

6.2.3.1 Manual start of Abaqus

Follow these steps:

- Place the file **`TwoValves.xml`** into the directory, in which the file **`TwoValves.inp`** locates.
- Open the *Windows*' command line or *Linux*'s terminal and navigate to the directory with the project.
- Start the *CSE Director* program, which connects *Abaqus* and *FlowVision*. Use the command:

  ```
  call abq2019 cse -config TwoValves -listenerport 5555
  ```

  The **`cse.log`** file will appear in the directory with the project. This file contains information that *CSE Director* has begun its operation.
- Start *Abaqus* using the command:

  ```
  abq2019 -job TwoValves ask_delete -csedirector localhost:5555
  ```

The **`TwoValves.log`** file will appear in the directory with the project. This wile contains information about start of *Abaqus* and connecting *Abaqus* to *FlowVision*.

6.2.3.2 Starting the computation from FlowVision

The project's computation is started from **Pre-Postprocessor**, see description of a standard starting in the section **Detailed description of a simplest model > Laminar flow in a tube > Starting the computation**.

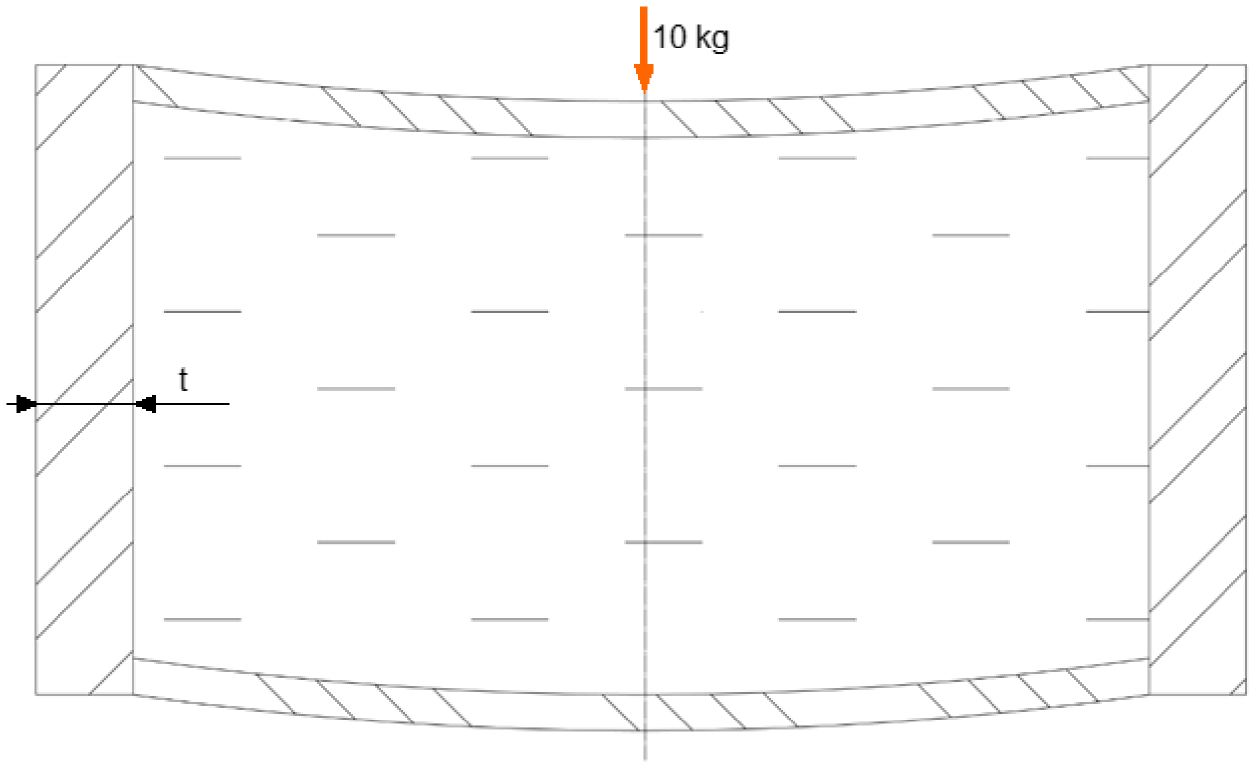
Stopping after 20 seconds of the simulated time is set in *Abaqus*. Also you can stop the project using the (**Stop computation**) button, and then resume the computation after editing the project.

6.3 Use of inverted geometry and tuning the artificial compressibility

Here you will become familiar with a simulation that uses an inverted **Moving body**.

Problem setting

In this exercise the program simulates a hydraulic damper, which is capable to damp oscillations arising in an external device or an assembly. The damper is a steel cylinder, wall of which is as thick as *t*, and superelastic diaphragms at bases. The interior of the damper is filled with water.



The steel shell is rigidly fixed while movement of the superelastic diaphragms is not limited. A load of 10kg is placed on the top surface of the upper superelastic diaphragms. Placing the load causes damped oscillations in the damper.

External boundaries of the computational domain will be changing during the simulation. *FlowVision* implements this behavior via an inverted **Moving body** modifier, which walls move according to the joint computation of *Abaqus* and *FlowVision* with replacing the geometry. The reversed geometry of the **Moving body** allows the program to use the computational domain inside the **Moving body**, not outside of it.

Coupling between *Abaqus* and *FlowVision* allows estimating of interaction a moving (or deforming) body and an internal or external flow.

Coupling simulations can be implemented by the following approaches:

- Monolithic approach: equations of hydroaerodynamics and equations for strength design are calculated by the same solver (a single one)
- Partitioned approach: equations are solved by two different solvers; some equations are solved by one solver while other equations are solved by the other solver. This approach is implemented in the *FlowVision-Abaqus* coupling.

In the monolithic approach interaction between the fluid and the structure on their boundary is considered synchronously. This approach allows keeping the energy being conservative in the system that improves numerical stability of the simulation. But such approach is focused on solving some specific type of problems. When the monolithic approach is used, tasks with other conditions and tasks of other types require modifying or re-engineering the solver and additional investigations.

So most of the tasks require specialized solvers, for example, rigidity of shells is simulated by a strength design solver while a hydrodynamic solver calculates an internal stream in the interior of the shells. Generally this interaction between the two solvers is a sequential process, which takes into account modular structure of the software and interrelations between the programs (such as *FlowVision* and *Abaqus*). In the partitioned approach equations for hydroaerodynamics and for structure are integrated alternatively and conditions for the interaction are not synchronized. This increases energy in the system and these schemes are often not not stable. Reducing the numerical instability is reached by decreasing the time step and/or by use of damping coefficients.

Improving stability of the computation due to adding the artificial compressibility in FlowVision

The artificial compressibility is added to reduce instability near a deformable wall specified in a CFD code. The artificial compressibility prevents sharp changes of pressure that can cause instability of the computation.

The equation

$$V_w^{n+1} = V_w^n + \frac{\partial P}{\partial t}(C + \Delta t^2 B),$$

where

$C = dl / dP$ is the flexibility

$B = A_w / m$ is the mobility

shows that the instability appears due to changing velocity of the wall, that is using for calculating the Navier-Stokes equations.

The artificial compressibility is specified by two coefficients, flexibility and mobility. Both the coefficients can be used either together or separately.

The material's flexibility characterizes the ability of a structure (wall) to move under applied pressure. Use of this setting is reasonable when the time step changes slightly during the simulation. In this case the initially specified coefficient will work stable during the computation.

The material's mobility characterizes acceleration of the wall under applied pressure and depends strongly on changing the time step during the computation. Use of this setting is effective when the time step changes substantially (by a decade or more) during the computation.

Formulae for these coefficients give the initial estimate of the artificial compressibility. This estimate can be excessive (cause a substantial change of weight or heavy damping) or can be inadequate (cause unstable simulation) and require additional assessment by tuning the coefficients. The value might change in the range of 1-3 orders.

Files:

Geometry model	`CylFSI.wrl`
Project in *Abaqus*	`CylFSI.inp`
Project in *FlowVision*	`CylFSI`

6.3.1 Preparing the project in FlowVision

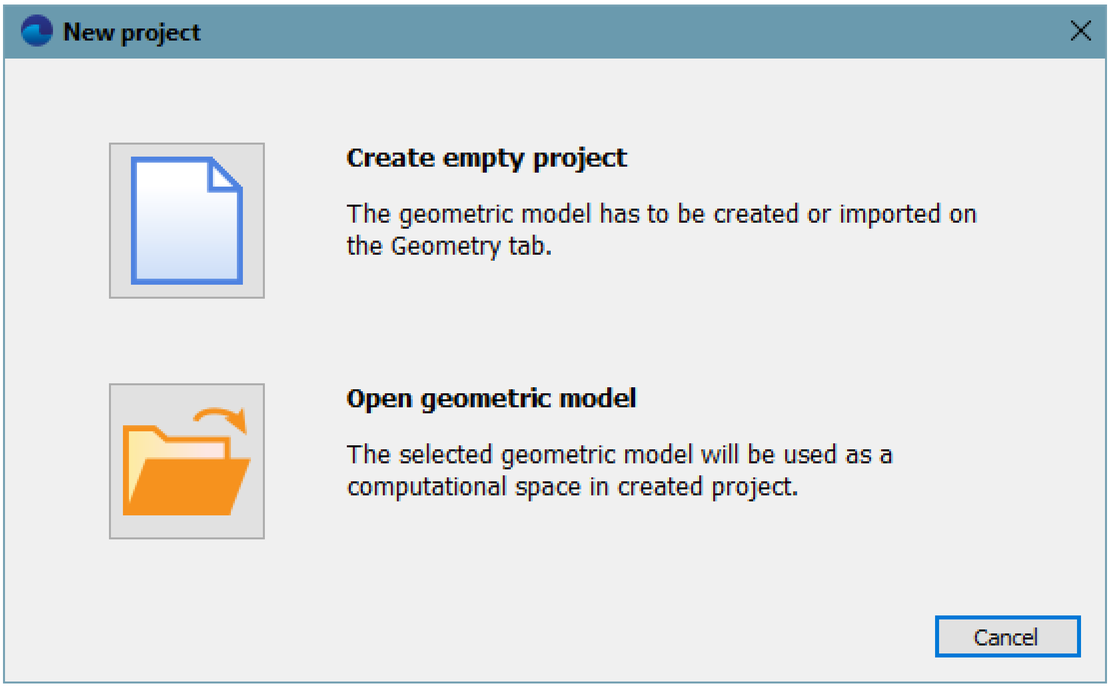
Follow the steps described in the sections below:

- Creating an auxiliary external subregion
- Physical model
- Creating an Imported Object and a «Moving body» modifier
- Boundary conditions
- Computational grid
- Parameters of FlowVision's calculation
- Visualization

6.3.1.1 Creating an auxiliary external subregion

FlowVision allows creating simple geometry for regions without use of external software. Creation a **Cylinder** object is enough for this exercise.

Open **Pre-Postprocessor**, apply the **File > Create** command from the main menu, and in the **New project** dialog box, which opens, select **Create empty project**:



The **Cylinder**, which is to be created, will be used as a region, into which an inverted **Moving body** will be placed. This means that the computational domain will locate not in the **Cylinder** but inside the **Moving body**. As the **Moving body** will change its shape during the computation, size of the **Cylinder** is to be enough for this movability.

In the **Geometry** tab apply the **Create** command from the context menu of the **Initial geom. models** folder to create the **Cone/cylinder #0** object and specify the following parameters in its properties:

Object > Location

- **Reference point**
  - **X** = 0
  - **Y** = -0.05
  - **Z** = 0
- **Axis X**
  - **X** = 0
  - **Y** = 1
  - **Z** = 0
- **Axis Y**
  - **X** = 0
  - **Y** = 0
  - **Z** = 1

Object > Parameters

- **Height** = 0.1
- **Radius 1** = 0.05
- **Radius 2** = 0.05
- **Base ratio** = 1

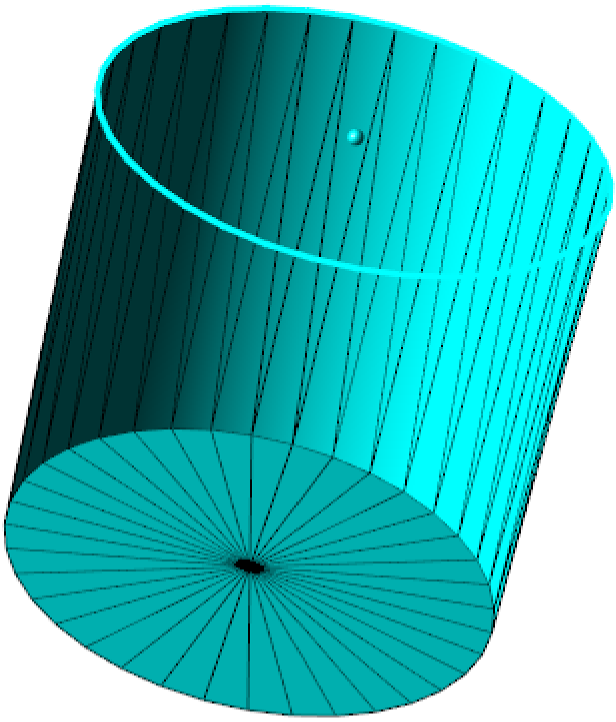
From the context menu of **Cone/cylinder #0** select **Create consistent mesh**. In the project tree the **Cone/cylinder #0** element will obtain a child element **mesh**.

From the context menu of the created consistent **mesh** select **Use in SubRegion Composer**. The **Cone/cylinder #0 - mesh** element will appear in the subfolder **SubRegion Composer > Objects**.

From the context menu of the element **SubRegion Composer > Composed subregions** select **Compose**. The **Cone/cylinder #0 - mesh** element will appear in the subfolder **SubRegion Composer > Composed subregions**.

After the composing, the object **Cone/cylinder #0 - mesh** can be used as the **Region's** geometry in the **Preprocessor** tab. From the context menu of the folder **Composed subregions** select **Use as Region main geometry**.

The **Preprocessor** tab will open, where the **Cone/cylinder #0 - mesh** subregion will appear in the **Subregions** folder.



6.3.1.2 Physical model

Specify parameters of the physical model in the **Preprocessor** tab.

In this exercise, the **Substance** is water. Properties of the **Substance** will be loaded from the standard **Substance database**.

- Open the context menu of the folder **Substances** and, using the **Create** command, create **Substance #0**.
- From the context menu of **Substance #0** select the command **Load from SD > Standard** and load the substance **Water** in its liquid phase. **Substance #0** will change its name to **Water_Liquid** and its child elements (**Molar mass**, **Density**, **Viscosity**, **Thermal conductivity**, ...) will receive their values from the **Substance database**.
- In properties of the element **Water_Liquid > Density** specify:

Value	**= 1000** (specify the value by a constant)
dRho/dP	**= 1e-20** (this parameter will be available after the density's value is specified by a constant)

In the folder **Phases**:

- Create a continuous **Phase #0**.
- In **Phase #0** add **Water_Liquid** into the subfolder **Phase #0 > Substances**.
- In properties of the folder **Phase #0 > Physical processes** specify:

Motion = **Navier-Stokes model**

In the folder **Models**:

- Create **Model #0**.
- Add **Phase #0** into subfolder **Model #0 > Phases**.

In properties of the element **Subregions > Cone/cylinder #0 - mesh** specify:

- **Model = Model #0**

6.3.1.3 Creating an Imported Object and a «Moving body» modifier

In the **Preprocessor** tab, from the context menu of the **Objects** folder, select **Create**.

The **Create new object** dialog box will open. Specify there **Object type = Imported object** and click **OK**.

Then select the file `CylFSI.wrl`, as the file, which contains geometry of the new **Imported object**.

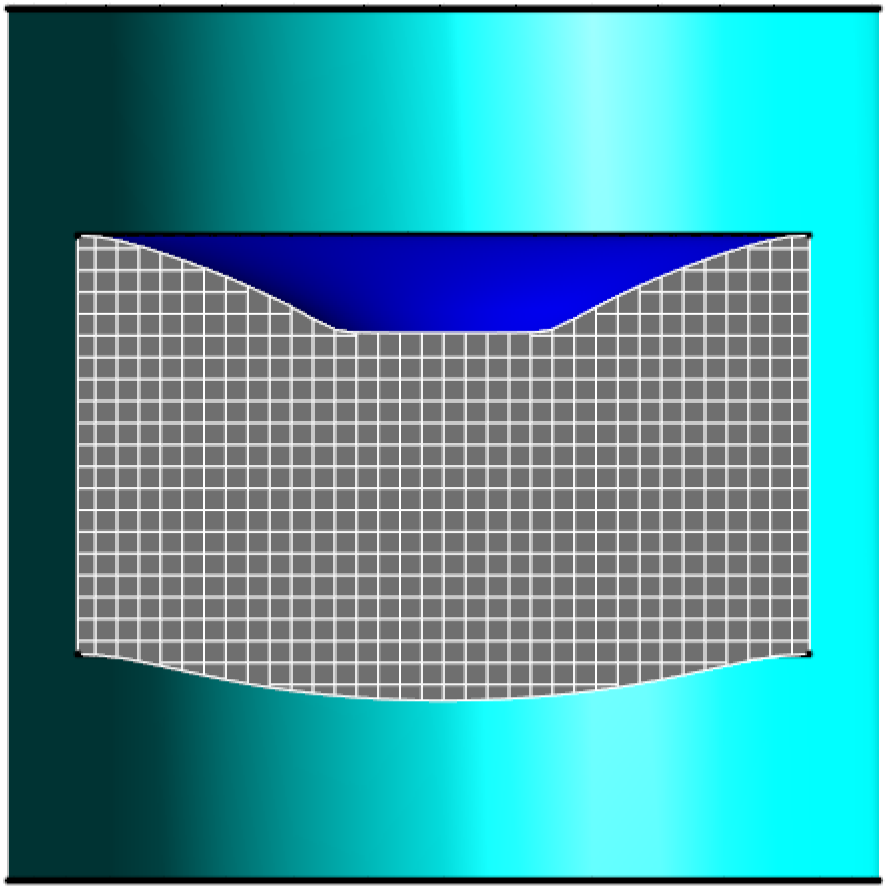
The program will create **Imported object #0**.

In the folder **Subregions > Cone/cylinder #0 - mesh > Modifiers** create a **Moving body** modifier; specify in the **Create new modifier**:
Modifier type = Moving Body
Objects = Imported object #0

The program will create **Moving Body #0**.

From the context menu of **Moving Body #0** select **Turn inside out**.

When the inverted geometry used, the computational domain is created within a **Moving body**.



Please notice parameters **FSI > Artificial compressibility**, **FSI > Flexibility** and **FSI > Mobility** in properties of **Moving Body #0**.

When you specify the artificial compressibility, you have to set **Artificial compressibility = Yes** and numerical values of parameters **Flexibility** and **Mobility**.

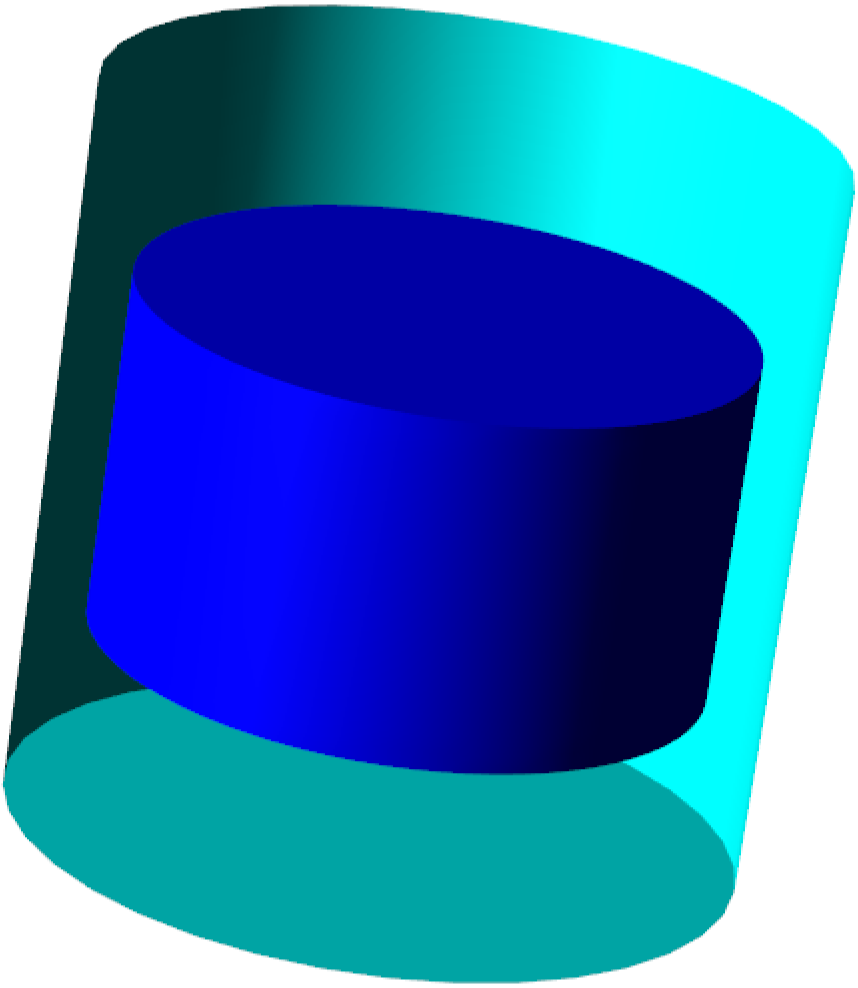
6.3.1.4 Boundary conditions

In this exercise the project contains two boundary conditions and they both are walls. The boundary conditions was created automatically after carrying out the following actions:

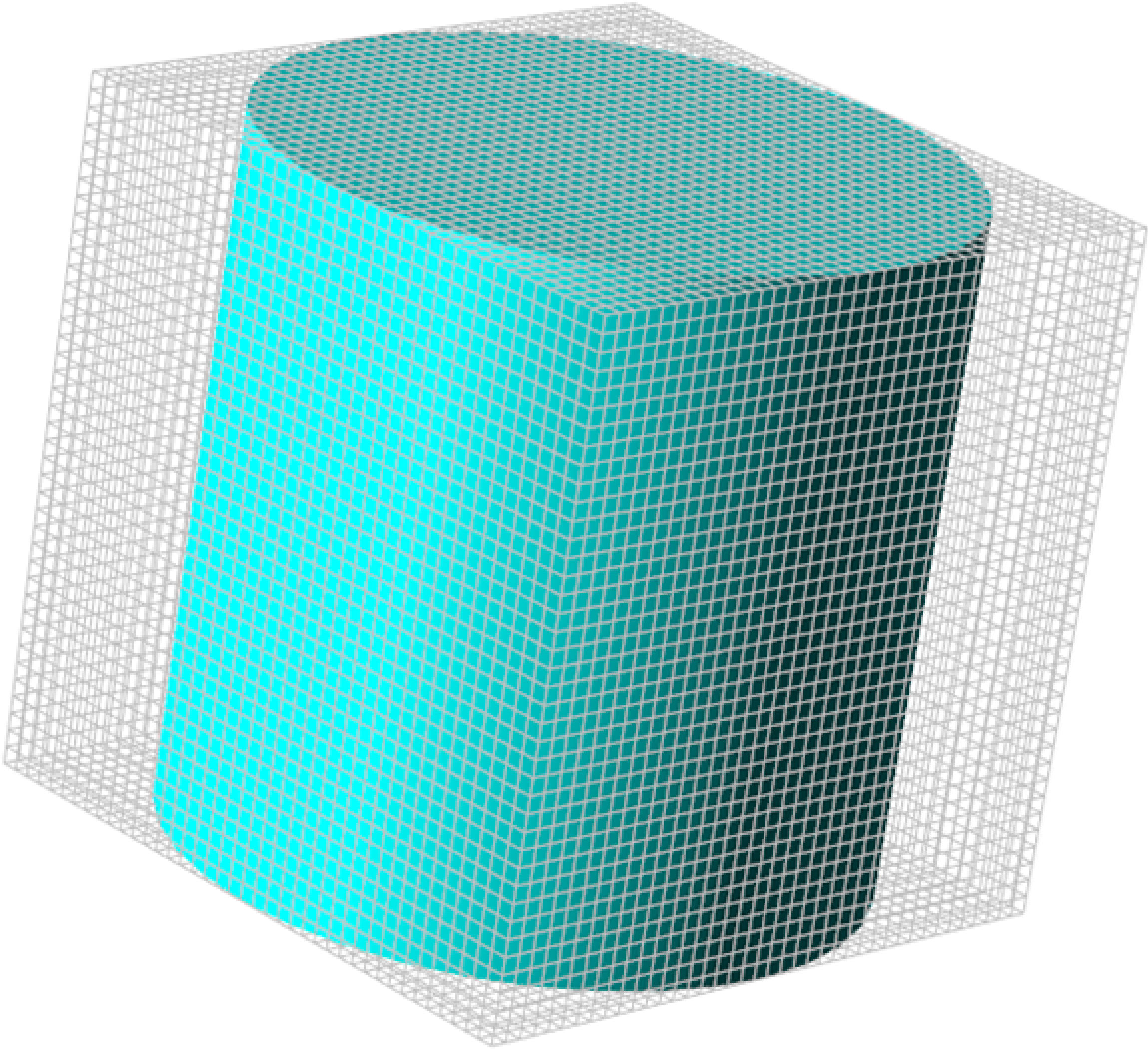
- The first boundary condition (**B.Cond. #0**, which is aqua) was created automatically at creation the subregion **Cone/cylinder #0 - mesh**.
- The second boundary condition (**B.Cond. #1**, which is blue) was created automatically at adding the **Moving Body #0** modifier into the project.

So the boundary conditions are already created.

Before the joint computation the model looks like this:



6.3.1.5 Computational grid



In properties of the **Initial grid** specify:
nX = 40
nY = 40
nZ = 40

In the **Properties** window of the **Initial grid** click **Apply**.

6.3.1.6 Parameters of co-simulation

From the context menu of the folder **External connections** select the **Create** command to create a new connector of the **Abaqus CSE** type. A dialog box will open where you have to select an **`inp`**-file; select there the file **`CylFSI.inp`**.

In properties of the folder **External connections** specify:
Exchange step > Method = FlowVision
Exchange step > Default time step = 0.001
Exchange step > Coef. for time step = 1
Old offset = Yes

In properties of the element **External connections > Abaqus CSE** specify:
Activation = Yes
Abaqus > Run ABAQUS = No
Abaqus > IP Source = User
Abaqus > Address = host name or IP address of the computer, on which *Abaqus* will run.

Abaqus > Port = 5555
Abaqus > Timeout [c] = 30
Loads relaxation > Scale factor = 1
Loads relaxation > Start in steps = 0
Loads relaxation > End in steps = 0
Loads relaxation > Initial coefficient = 1
Loads relaxation > Final coefficient = 1
Heat relaxation > Scale factor = 1
Heat relaxation > Start in steps = 0
Heat relaxation > End in steps = 0
Heat relaxation > Initial coefficient = 1
Heat relaxation > Final coefficient = 1

In properties of the element **ASSEMBLY_EXSURF** specify: **Moving body = Moving body #0**.

Do not change other settings.

6.3.1.7 Generating the CSE Director's configuration file

To generate the *CSE Director*'s configuration `xml`-file, follow these steps:

In the **Properties** window of the **External Connections > Abaqus CSE** folder click the **Operations >** **(Save configuration file for CS-service)** button. The **Export to configuration file** dialog box will open. Specify parameters there:
Cosolution scheme = Gauss-Seidel with prediction
Exchange Step = Let master program choose
Master program = FlowVision
Time stopping criteria = 20

Click **OK**. In the directory, which contains the client part of the project, the `CylFSI.xml` file will appear.

6.3.1.8 Parameters of FlowVision's calculation

The joint computation requires limiting the time step. As the body doesn't move at initial iterations, *FlowVision* will start the computation with the default time step, which is equal to **1** [s]. Accordingly, the exchange step with the FSI software will be also **1** and exchange between the programs will not begin for a long time.

To fix this issue, you have to limit the time step for the initial 10-100 iterations. Value of the maximal time step is recommend to be set the same as the value of the initial step in *Abaqus*.

In the **Solver** tab, in properties of the **Time step** element, specify:

Method	**= Via CFL number**
Convective CFL	**= 20**
Surface CFL	**= 1**
Max step	**= 0.001**

6.3.1.9 Visualization

Open the **Postprocessor** tab.

From the context menu of **3D-scene > Objects > Plane #0** select **Clipping object**.

From the context menu of the folder **3D-scene > Objects > Computational space > Solids** select **Apply clipping**.

Then set the visualization as described in subsections below.

Variation of volume, maximal pressure, and minimal pressure

Open the **Preprocessor** tab.

In the **Characteristics** folder create **Characteristics** on the **Computational space**.

In properties of the just created **Characteristics #0 (Computational space)** specify **Variable > Variable = Pressure**.

Open the **Solver** tab.

From the context menu of the folder **Stopping conditions > User values** select **Create** to create **Stop criterion #0**.

In properties of the just created **Stop criterion #0** specify:
Name = Volume
Object = Characteristics #0 (Computational space)
Variable = Volume (this parameter will be available when you specify the **Object** parameter above)

Create another **Stop criterion** with the following properties:
Name = Maximal pressure
Object = Characteristics #0 (Computational space)
Variable = Maximum (this parameter will be available when you specify the **Object** parameter above)

Create the third **Stop criterion** with the following properties:
Name = Minimal pressure
Object = Characteristics #0 (Computational space)
Variable = Minimum (this parameter will be available when you specify the **Object** parameter above)

Variations of the variables, which were used for specifying the **Stop criteria**, will be displayed during the computation in the **Plot** tab of the **Monitoring** window.

Pressure distribution

Open the **Postprocessor** tab.

From the context menu of the folder **Layers** select **Create**, and in the dialog box **Create new layer** specify:
Layer type = Color contours
Objects = Plane #0

In properties of the new **Layer** specify:
Name = Pressure, Pa
Variable > Variable = Pressure
Value range > Mode = Manual
Value range > Max = 2000
Value range > Min = -500
Palette > Appearance > Enabled = Yes
Palette > Appearance > Title = Yes
Palette > Appearance > Style = Style 1
Palette > Appearance > Color = Black

Velocity distribution

From the context menu of the folder **Layers** select **Create**, and in the dialog box **Create new layer** specify:
Layer type = Vectors
Objects = Plane #0

In properties of the new **Layer** specify:
Name = Velocity, m/s
Variable > Variable = Velocity
On regular grid = No
Coloring > Variable > Variable = Velocity
Coloring > Value range > Mode = Manual
Coloring > Value range > Max = 1
Coloring > Value range > Min = 0
Coloring > Palette > Appearance > Enabled = Yes
Coloring > Palette > Appearance > Title = Yes
Coloring > Palette > Appearance > Style = Style 1
Coloring > Palette > Appearance > Color = Black

Cross-section of the computational grid

Cross-section of the computational grid visualizes the area, where the computation will run.

From the context menu of the folder **Layers** select **Create**, and in the dialog box **Create new layer** specify:
Layer type = Computational grid section
Objects = Plane #0

6.3.2 Start of joint computation

For this exercise it is possible to start a joint computation using **MPM-Agent** (in *FlowVision 3.12.04* or newer versions). In older *FlowVision* versions only manual start of a joint computation is possible.

To run this exercise using **MPM-Agent**, apply settings described in the exercise Deformable valve in channel (see details in the section Starting and stopping the computation).

6.3.2.1 Manual start of Abaqus

Follow these steps:

- Place the file **`CylFSI.xml`** into the directory, in which the file **`CylFSI.inp`** locates.
- Open the *Windows*' command line or *Linux*'s terminal and navigate to the directory with the project.
- Start the *CSE Director* program, which connects *Abaqus* and *FlowVision*. Use the command:

  ```
  call abq2019 cse -config CylFSI -listenerport 5555
  ```

  The **`cse.log`** file will appear in the directory with the project. This file contains information that *CSE Director* has begun its operation.
- Start *Abaqus* using the command:

  ```
  abq2019 -job CylFSI ask_delete -csedirector localhost:5555
  ```

  The **`CylFSI.log`** file will appear in the directory with the project. This wile contains information about start of *Abaqus* and connecting *Abaqus* to *FlowVision*.

6.3.2.2 Starting the computation from FlowVision

The project's computation is started from **Pre-Postprocessor**, see description of a standard starting in the section **Detailed description of a simplest model > Laminar flow in a tube > Starting the computation**.

Stopping after 20 seconds of the simulated time is set in *Abaqus*. Also you can stop the project using the (**Stop computation**) button, and then resume the computation after editing the project.

Run the project. The project will end with crash of the solver after several exchanges. The crash of the solver is caused by a sharp growth of pressure in the enclosed volume and subsequent stop of the computation.

To fix this issue, you have to tune artificial compressibility coefficients that provide stable solution.

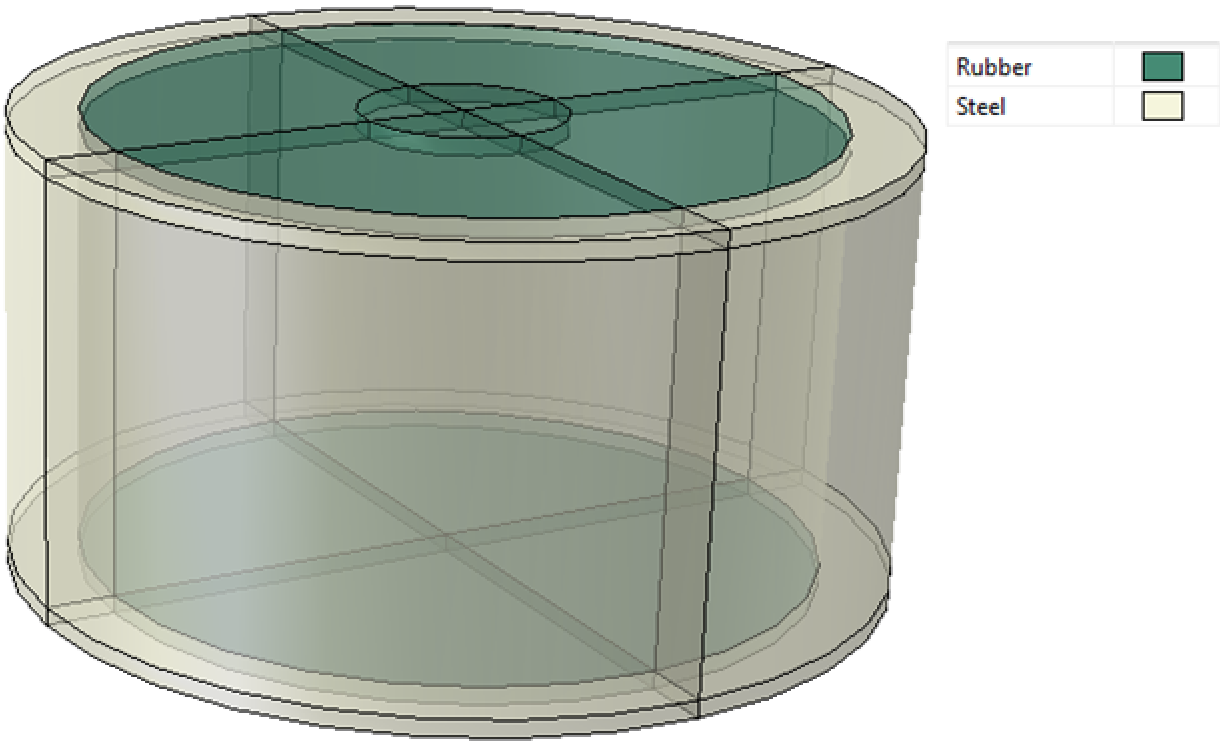
Before you begin to tune the artificial compressibility coefficients, specify in the **Preprocessor tab**, in properties of the element **Region > Substances > Water_Liquid > Density** specify **dRho/dP = 0** (here we change the previously set value dRho/dP=1e-20 with a zero value, which, at the enabled artificial compressibility, will not cause crash of the computation.

6.3.3 Tuning the artificial compressibility

6.3.3.1 Estimate of flexibility in Abaqus

The *Abaqus* model is a cylinder consisting of two materials (thickness of the walls corresponds to materials used).

Most flexible part in this model is a hyperelastic element. Deformation of steel will be too small even under large loadings. So it is reasonable to estimate flexibility of the hyperelastic material only.



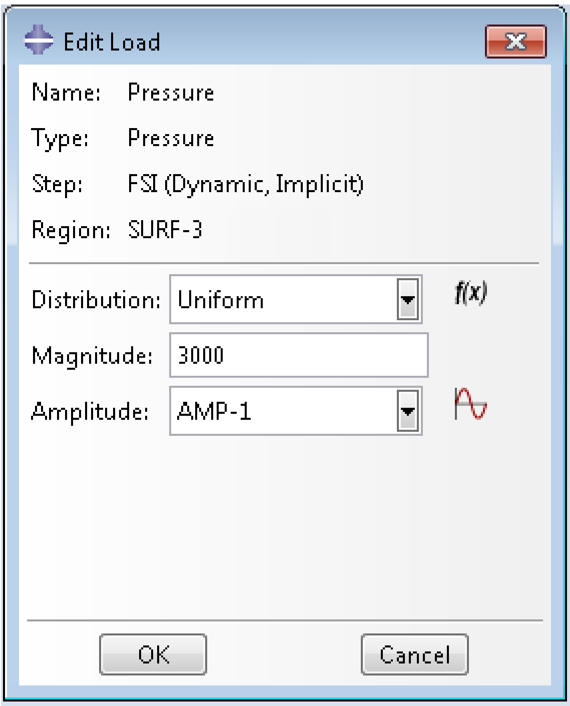
In *Abaqus/CAE* apply the command **File > Import > Model** and import the file **`CylFSI.inp`**.

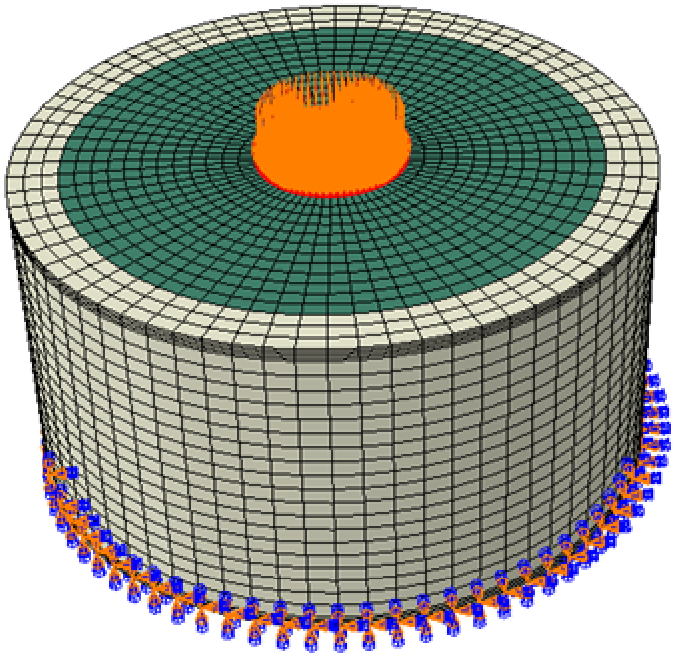
To estimate the flexibility, you have calculate change of the load and displacement under this load during a small time shift. When obtaining the coefficient, you have to use a small time step in the *Abaqus* project.

Navigate to the module **Steps** and specify the following parameters for the **FSI** step:
Basic > Time period = 1
Incrementation > Initial increment size = 1E-6
Incrementation > Maximum increment size = 1E-6

From the context menu of **Loads > Gravity** select **Suppress**. This load will be hidden from the analysis.

Instead of this load create a pressure and apply it to the surface as shown on the illustration below. **Magnitude** of the pressure is to be selected as to **3000** [Pa]. Also the existing value **AMP-1** is to be selected in the **Amplitude** field.

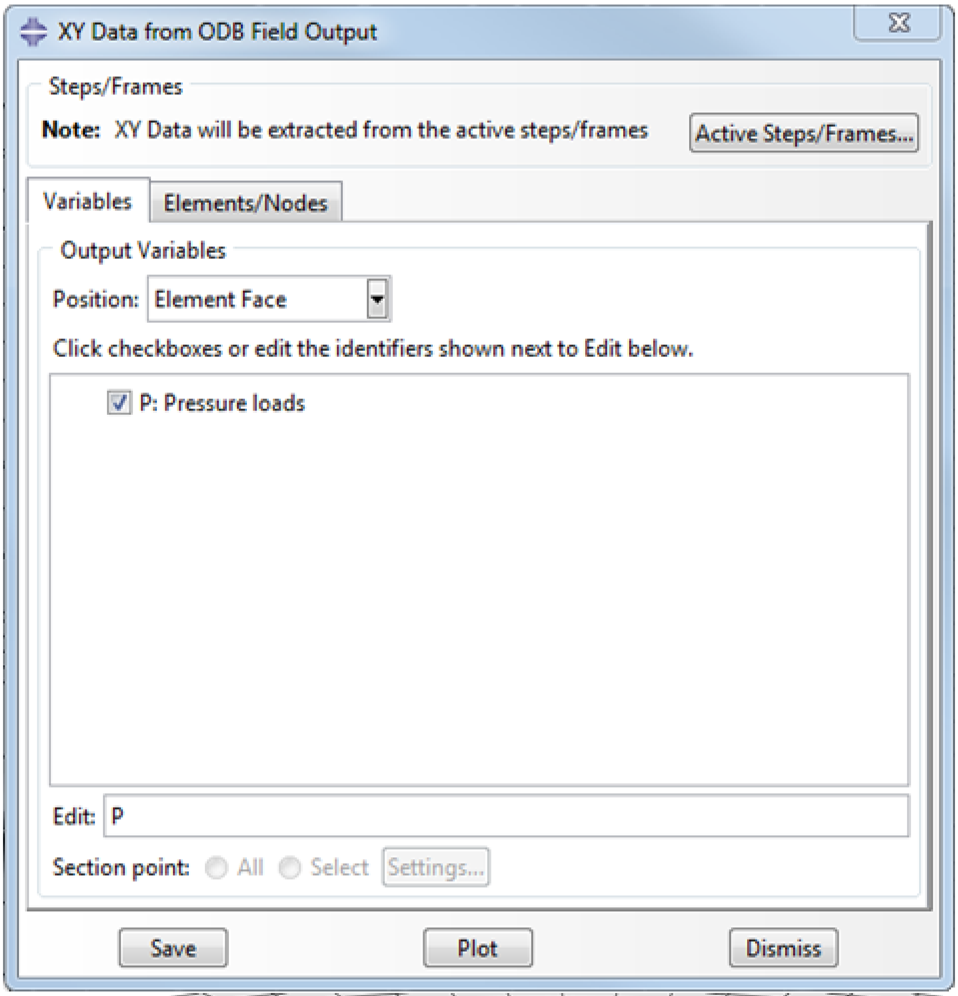


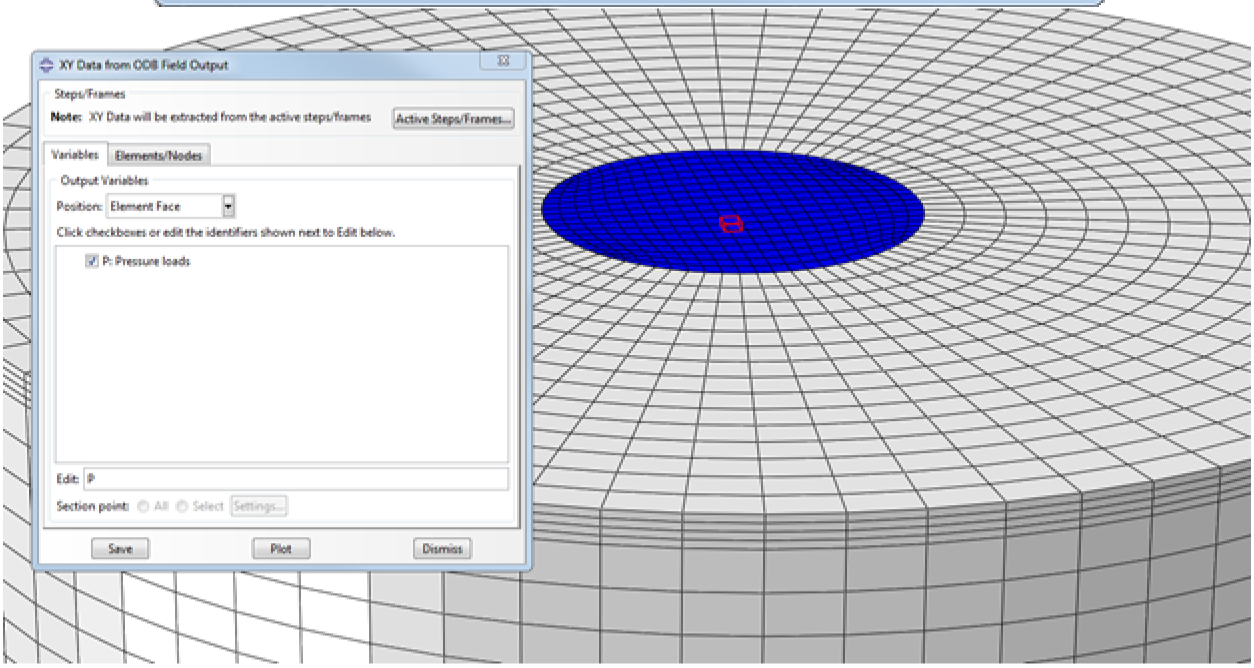


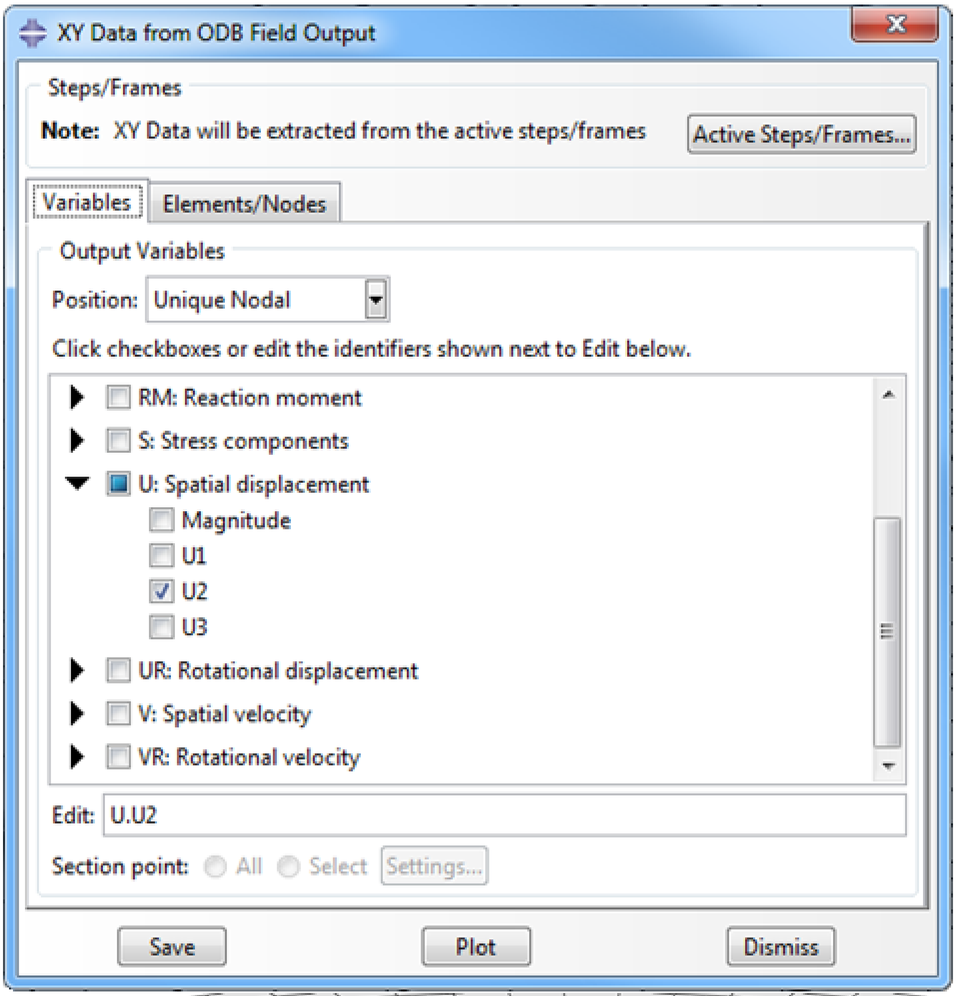
Run the simulation. It is not necessary to wait ending of the simulation, it is enough to wait for 30-50 iterations.

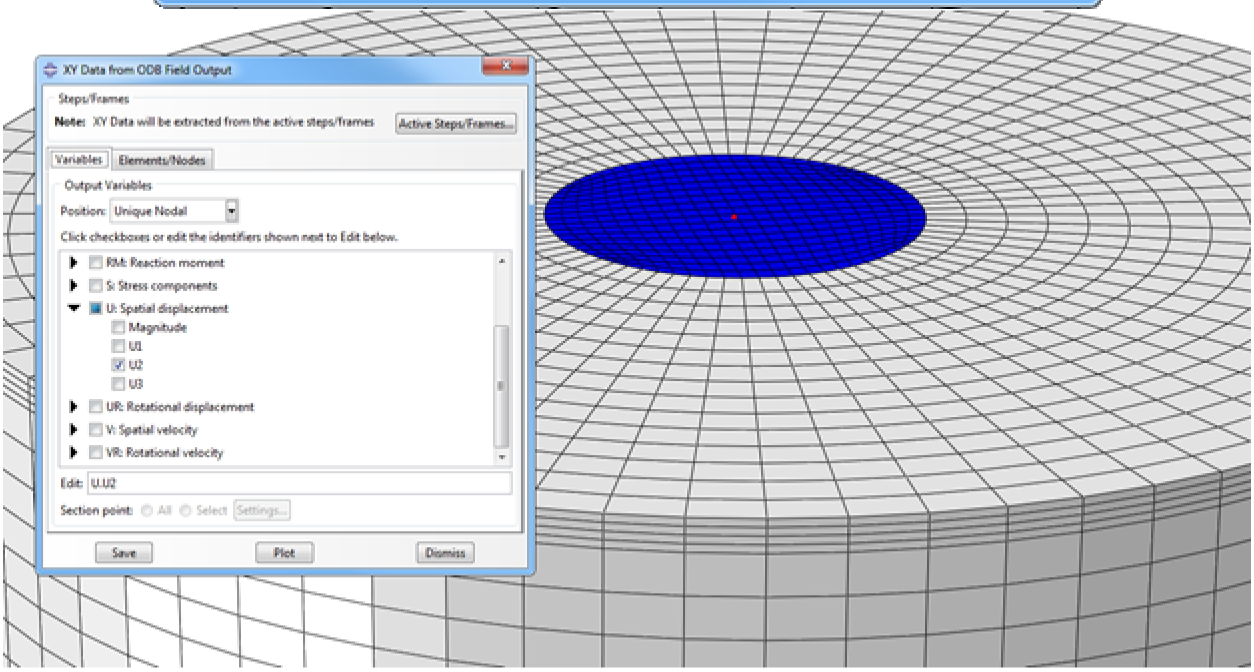
Open the *ODB*-file of the simulation. Use **Create XY Data > ODB Field Output** to create following plots:

- plot of pressure (output variable is **P**, position is **Element Face**)
- plot of displacement along axis Y (output variable is **U.U2**, position is **Unique Nodal**)

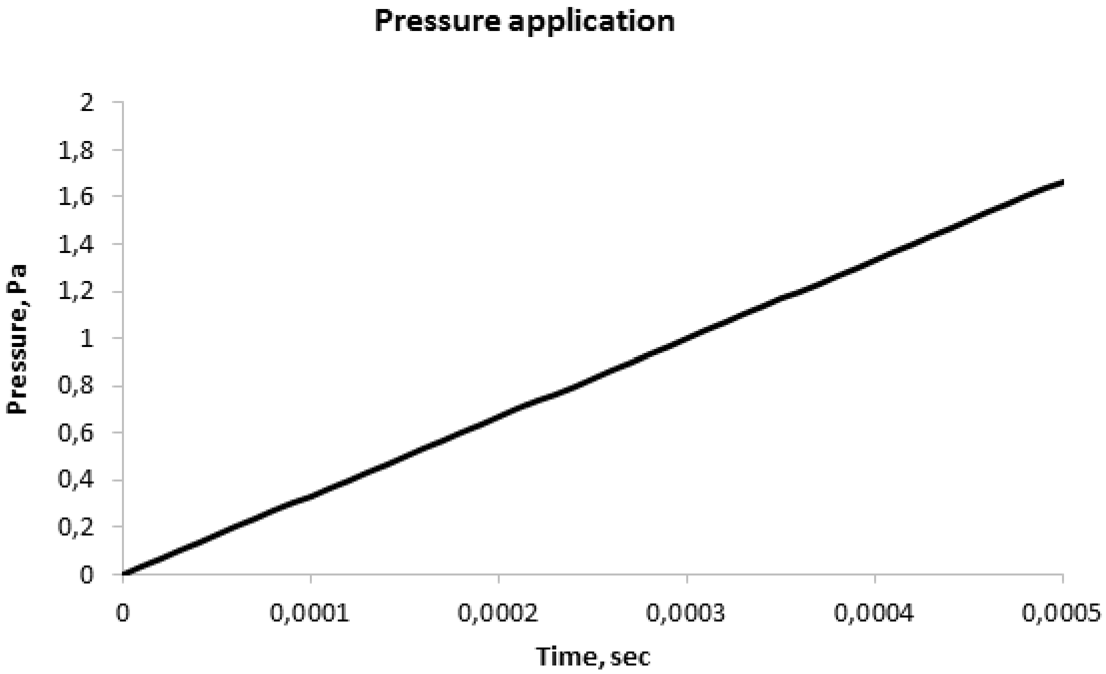

XY Data from ODB Field Output
Steps/Frames
Note: XY Data will be extracted from the active steps/frames
Active Steps/Frames...
Variables
Elements/Nodes
Output Variables
Position: Element Face
Click checkboxes or edit the identifiers shown next to Edit below.
P: Pressure loads
Edit: P
Section point: All Select Settings...
Save
Plot
Dismiss

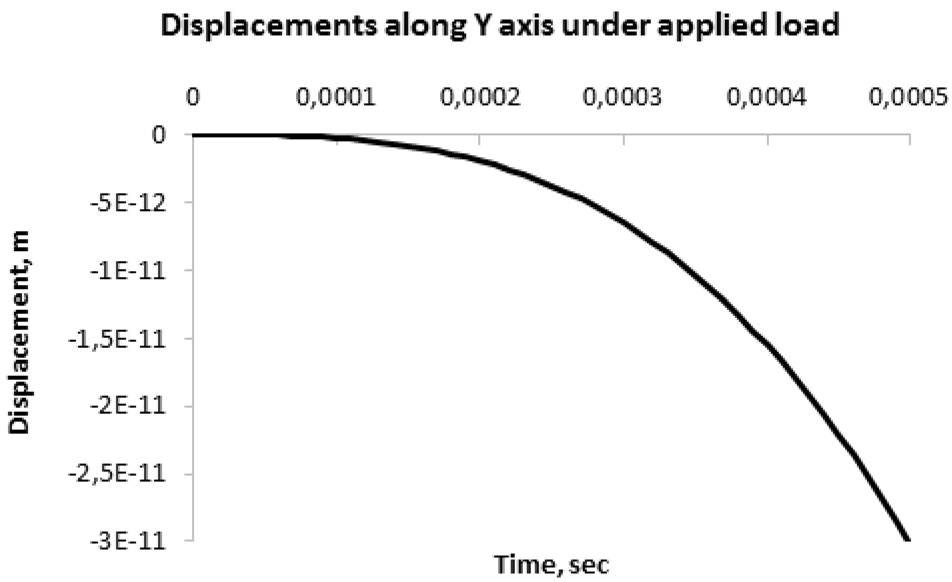






These plots can be exported to *Excel* by the command **Plugins > Tools > Excel Utilities**.

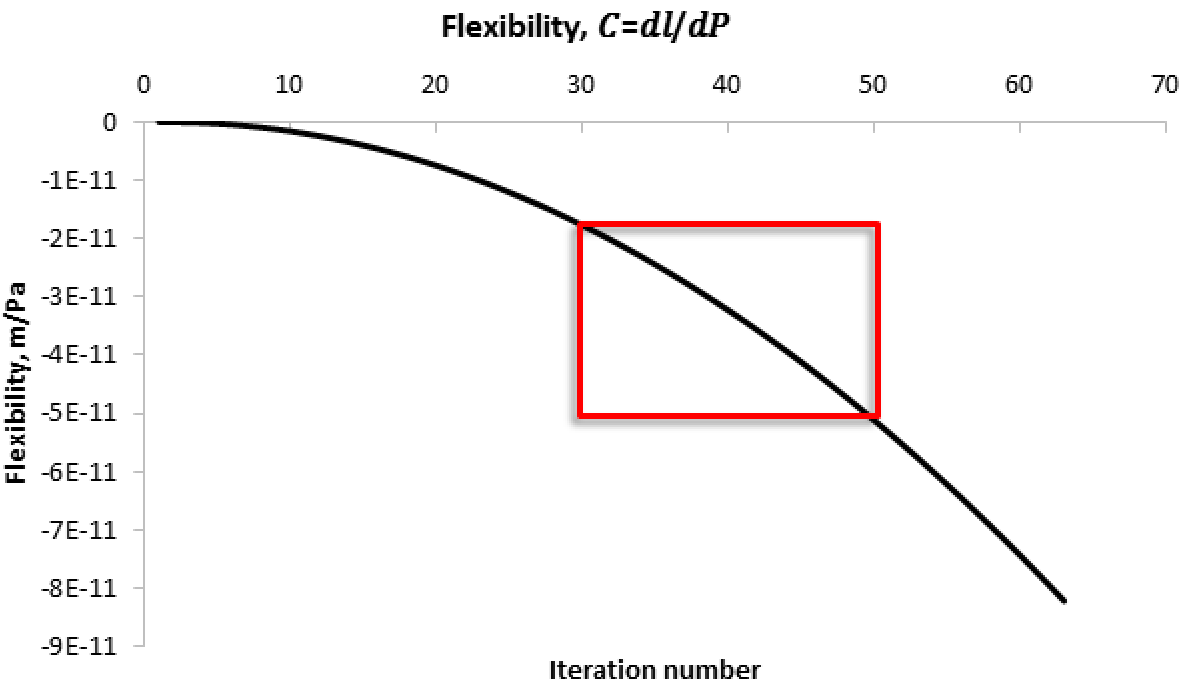




Use the obtained data to calculate the value of flexibility as ration of change of displacement to change of pressure.

The flexibility is calculated by the formula: $C=dl/dP$.

The flexibility is recommended to be calculated starting from 30th - 50th iteration because of too small pressure is applied at the beginning of the simulation:



The flexibility C is approximately equal to $5 \cdot 10^{-11}$ [m/Pa].

As in this case the displacement vector is directed against the Y axis, the absolute value of the calculated value of the flexibility is to be taken.

6.3.3.2 Estimate of mobility in FlowVision

To estimate the mobility, it necessary to know surface area and mass of the body, which will interact with the liquid.

Mass of the body (of the hyperelastic elements) can be received from *Abaqus/CAE*. Import the `CylFSI.inp` file.

To display the hyperelastic element, navigate to **Tools > Display Group > Create > Item: Sets** and select **PART-1-1.SET-2**. In the dialog box **Perform a Boolean on the viewport contents and the selection** click **Replace** and close the **Create Display Group** dialog box.

Navigate to the Mesh module. Use **Tools > Query > Mass Properties** and find the mass of the hyperelastic elements, which is **0.0511** [kg].

Area of the surface can be found in *FlowVision*. Use the **File > Preferences** command to open the **Preferences** dialog box and specify there **Display > Show all groups = Yes**. In the project tree below the element **Region > Subregion > Cone/cylinder #0 - mesh > Geometry > Moving body #0** find the group that matches to the surface of hyperelastic element and its properties look for its area, which is to be **0.0055** [m^2]. Area of both hyperelastic elements is to be **0.011** [m^2].

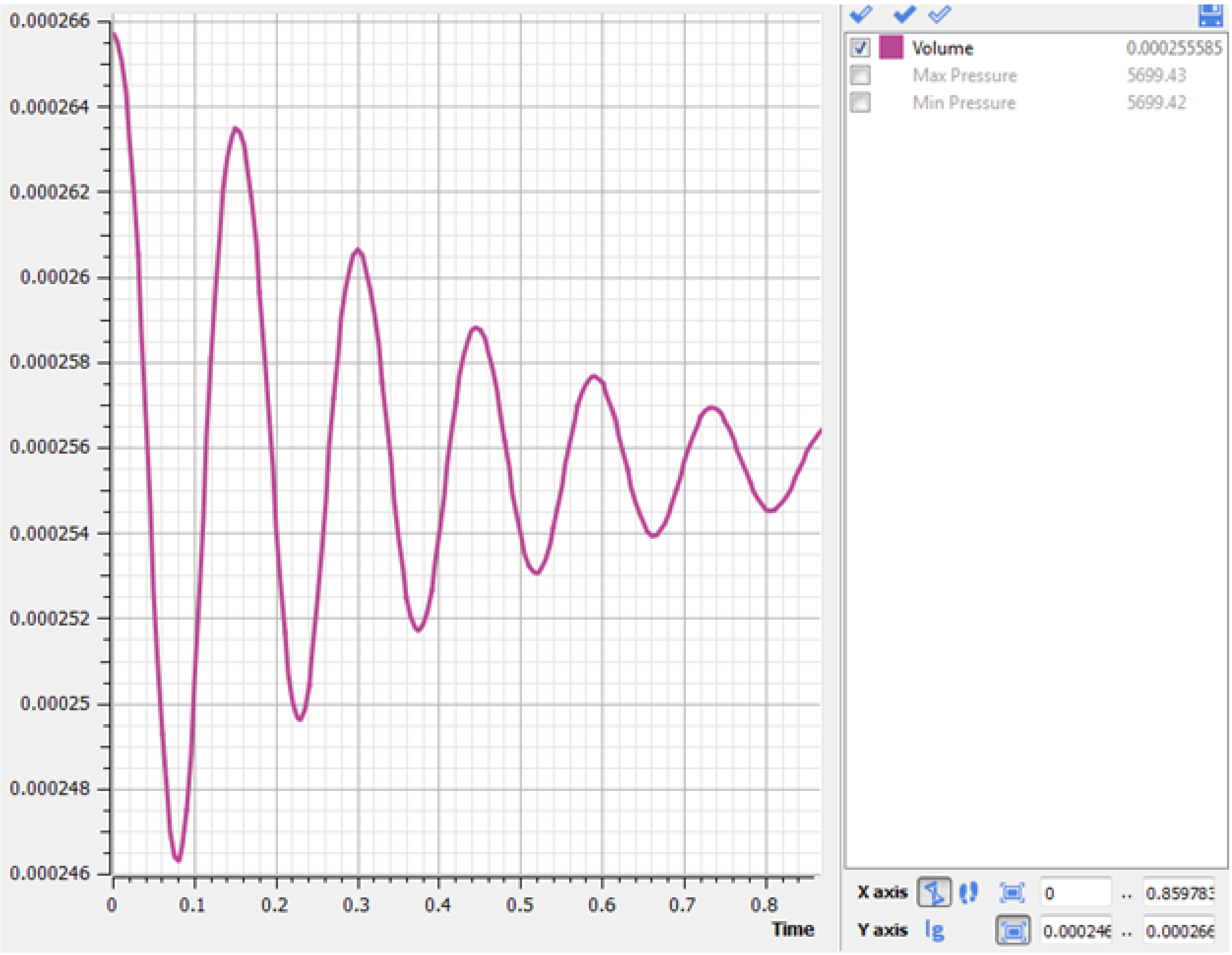
The estimate of mobility is calculated by the formula $B=A_w/m$ and is to be **0.21** [m^2/kg].

6.3.3.3 Investigation of artificial compressibility

Obtained estimates of flexibility and mobility are to be specified in the project tree in properties of the **Region > Subregion > Cone/cylinder #0 – mesh > Modifiers > Moving Body #0** element:

- **FSI > Artificial compressibility = Yes**
- **FSI > Flexibility = 5E-11**
- **FSI > Mobility = 0.21**

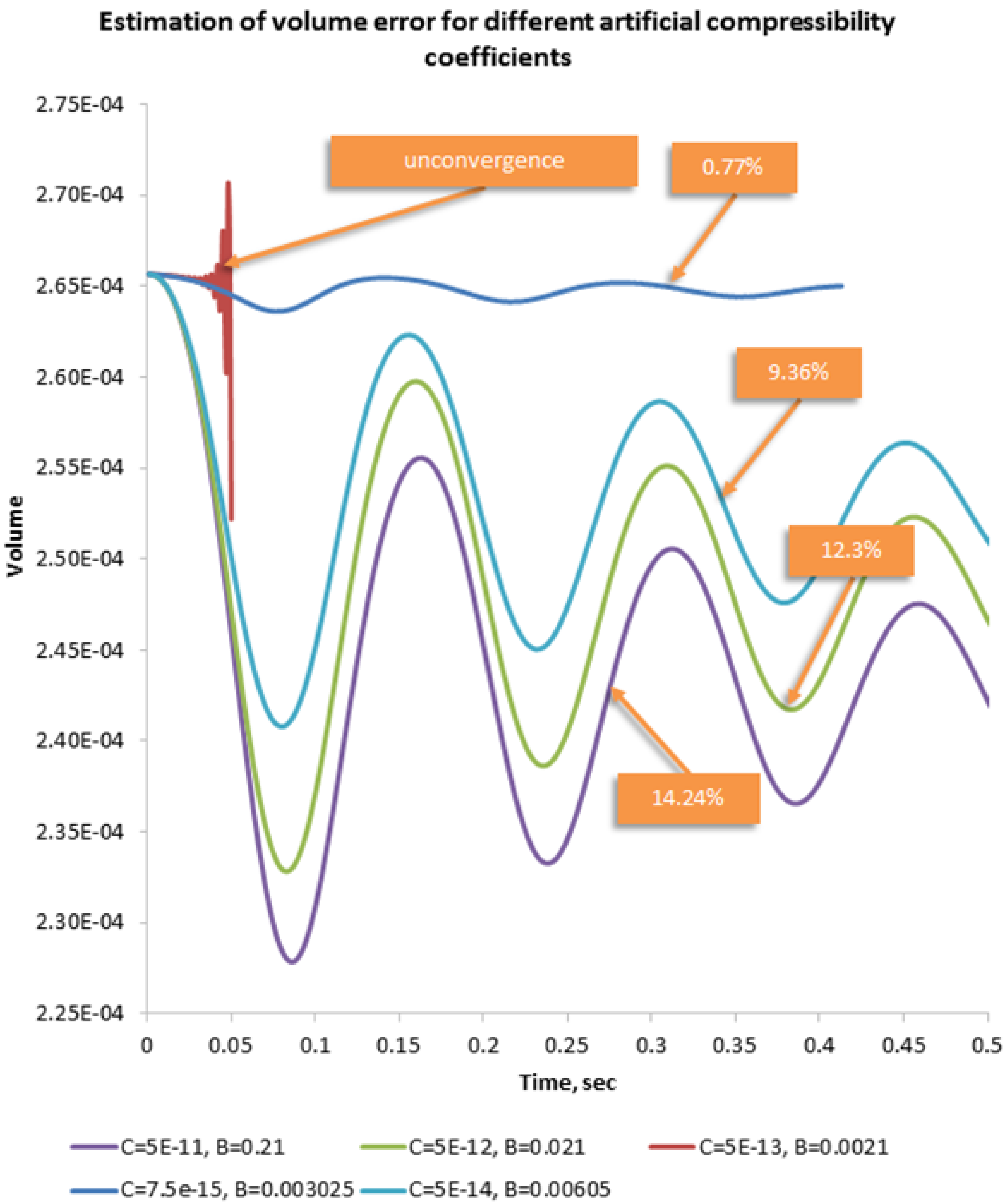
Run the joint simulation and wait until the volume begins to convergence:



The obtained plot can be used to evaluate the error caused by entering artificial compressibility into the computation.

Decreasing coefficients of the artificial compressibility can cause reducing the error but too small coefficients can cause nonconvergence of the simulation. To tune the coefficients correctly, you have to run the simulation several times for several iterations using different coefficients of artificial compressibility at each run (the coefficients should be increased/decreased by an order or twice/three times more/less).

Value of the error is to be estimated by the formula $\delta V=(V_{max}-V_{min})/V_0$.



After several trial runs with various coefficients the following combination is to be selected, which provides the computational error 0.77% (shown by the dark blue line on the plot):

- flexibility $C = 7.5 \cdot 10^{-15}$ [m/Pa]
- mobility $B = 0.003025$ [m^2/kg]

Use of these coefficients gives minimal errors and provides stable and accurate solution. Smaller coefficients will cause an unstable simulation. When the coefficients are too large, the artificial compressibility will substantially affect the solution, which will become unphysical.

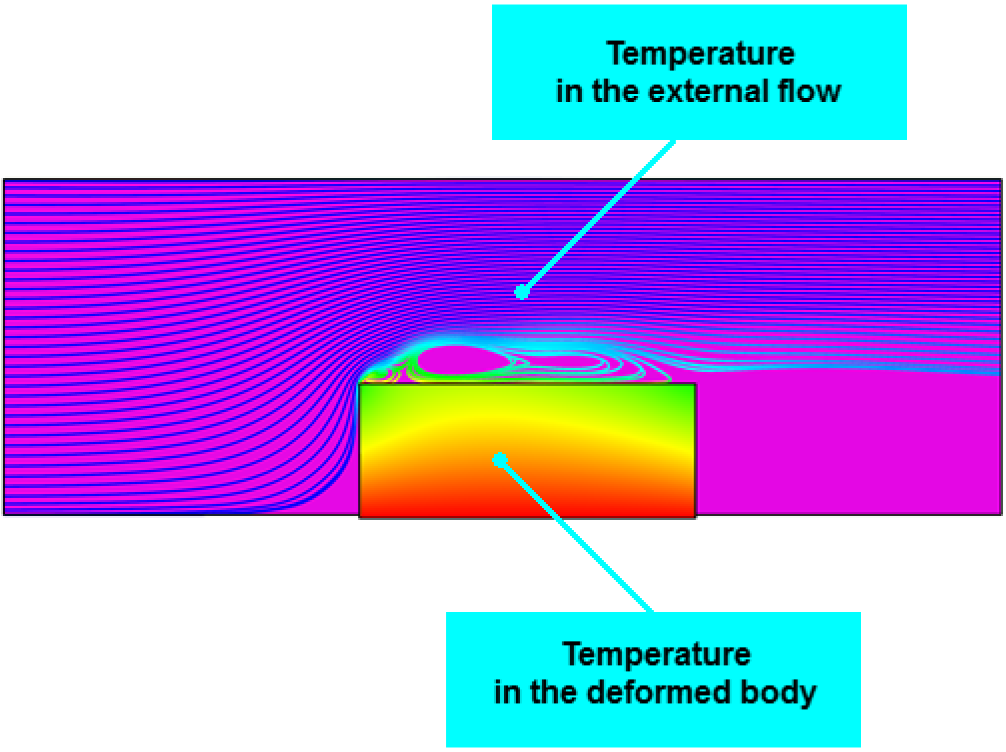
6.4 External heat exchange in the FlowVision-Abaqus conjunction

Problem setting

Solving the problem of heat transfer as a joint simulation with an third-party finite-element analysis (FEA) software allows you to evaluate deformation of a structure, which is under the influence of external heat flows. Also the FEA software allows you to take into account anisotropic properties of the material; this provides better evaluation of heat transfer in the deformed body.

The heating process will be simulated in *Abaqus*. Temperature of 100°C applied to the base of the brick, which is the deformed body, at the first step of the computation. The joint simulation of heat transfer on the brick's boundaries in the *FlowVision-Abaqus* conjunction starts at the second step of the computation. *FlowVision* simulates the forward flow heated by walls of the brick and dynamics of the temperature's field. *Abaqus* simulates heat transfer in the solid body and deformations caused by the temperature expansion.

As in the previous exercises, in this exercise we study creation of projects both in *Abaqus* and *FlowVision*, and also study their interrelations and starts. If you use a ready model **`BrickABQ.inp`**, you can skip the section about preparing the *Abaqus* project.



Files:

Project in *Abaqus*	**`BrickABQ.inp`**
Project in *FlowVision*	**`Brick`**

6.4.1 Preparing the project in Abaqus

Follow the steps described in the sections below:

- Preparing the geometry model of the brick in Abaqus
- Specifying an interface surface in Abaqus
- Specifying boundary conditions and loads in Abaqus
- Generating an inp-file
- Modifying the inp-file of the Abaqus' project

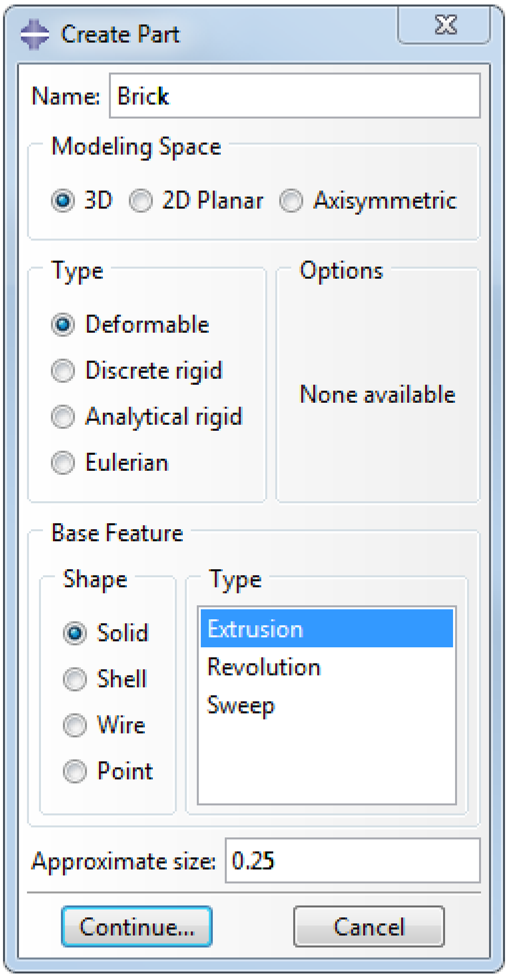
6.4.1.1 Preparing the geometry model of the brick in Abaqus

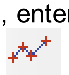
This section describes how you can create the model of a brick in *Abaqus* (version *2019*) by your own. You can skip this section if you use a ready model `BrickABQ.inp` and return to this section when you wish to receive appropriate experience.

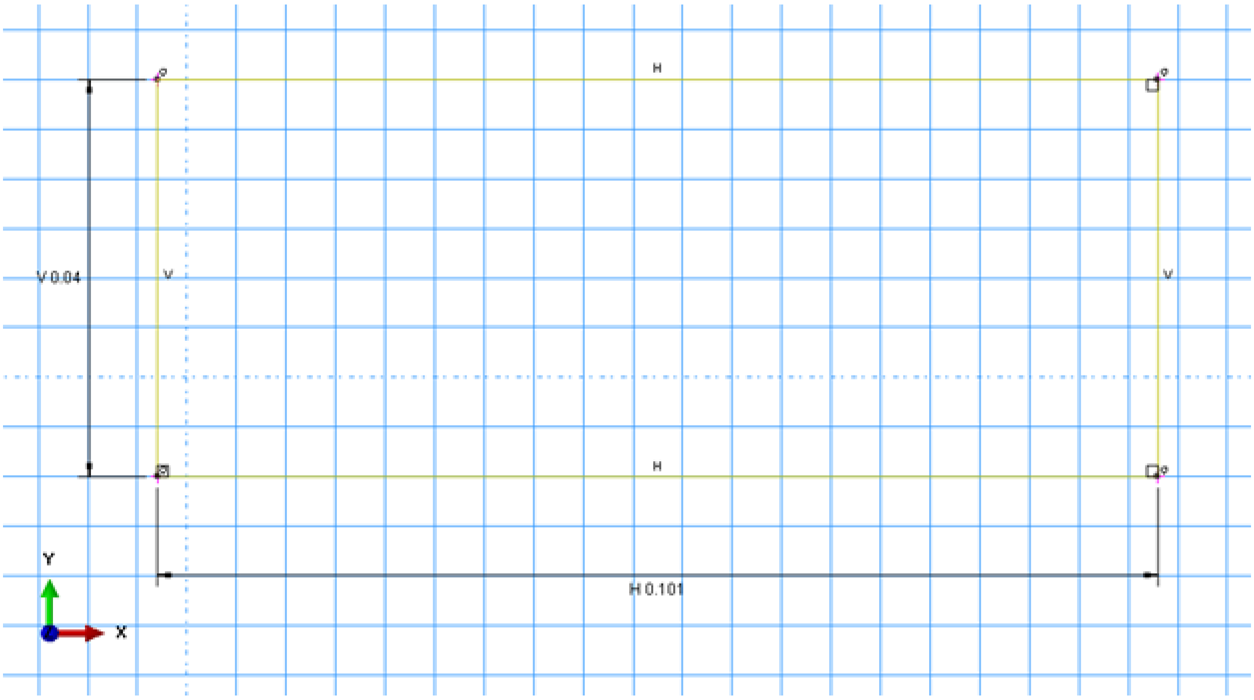
1. Open **Model > Edit Model Attributes > Model-1** and specify the absolute zero temperature:
 Absolute zero temperature = -273

2. Click the icon  (**Create Part**). The **Create Part** dialog box will open, specify there:
 Name = Brick
 Modeling space = 3D
 Type = Deformable
 Base Feature > Shape = Solid
 Base Feature > Type = Extrusion
 Approximate size = 0.25

 and click **Continue**.

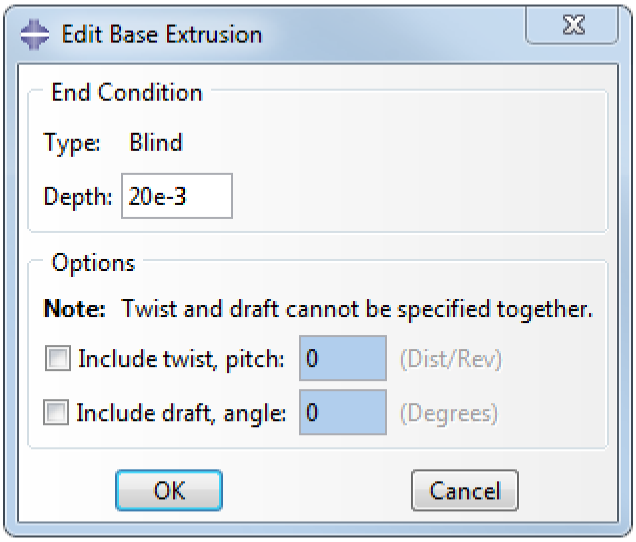


2. In the graphical work window, which opens, use the tool  (**Create Isolated Point**) to create points, which will be used to create the brick's contour. To do so, enter the following coordinates: **[-0.003,0.03];[-0.003,-0.01];[0.098,0.03];[0.098,-0.01]**. Then use the tool  (**Create Lines Connected**) to outline the contour by lines. To do so, click **Done** (the mouse wheel).

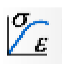


Making a contour for the part Brick

3. The **Edit Base Extrusion** dialog box will open. Enter the brick's thickness, enter the value **20e-3** into the **Depth** field and then click **OK**:



Creating the part Brick

4. Specify properties of the material. Navigate to the module **Property**. Click  (**Create Material**) to create a new material.

5. The **Edit Material** dialog box will open; specify there:

Name = Steel (name of the material)

General > Density = 7890 (density of the material, [kg·m^{-3}])

Mechanical > Elasticity = Elastic

Mechanical > Type = Isotropic

Mechanical > Young's Module = 2E11 (Young's modulus, [Pa])

Mechanical > Poisson's Ratio = 0.3 (Poisson's ratio)

Mechanical > Expansion > Type = Isotropic

Mechanical > Expansion > Expansion Coeff = 7.2E-6

Thermal > Conductivity > Type = Isotropic

Thermal > Conductivity > select the checkbox **Use temperature-dependent data**

Thermal > Conductivity > specify the data:

λ, [W/(m °C)]	T, [°C]
48.5	**0**
46.5	**100**
44	**200**
40.8	**300**

Thermal > Specific Heat > Type = Constant Volume

Thermal > Specific Heat > select the checkbox **Use temperature-dependent data**

Thermal > Specific Heat > Number of field variables = 0

Thermal > Specific Heat > specify the data:

C_p, [J/°C]	T, [°C]
466	**0**
486	**100**
507	**200**
523	**300**

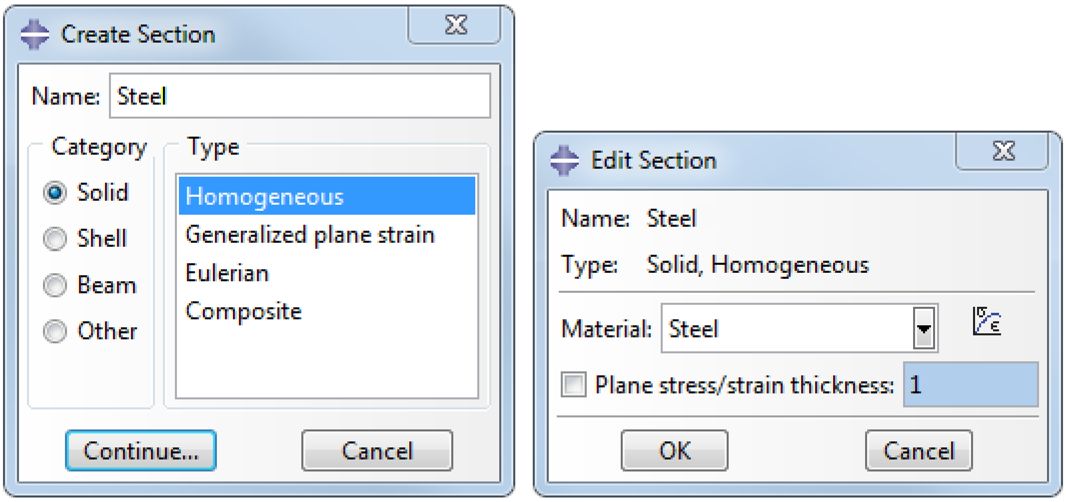
and click **OK**.

6. Create a section made of the material **Steel**. Click  to open the **Create Section** dialog box. Specify there:

Name = Steel (name of the section)
Category = Solid
Type = Homogeneous

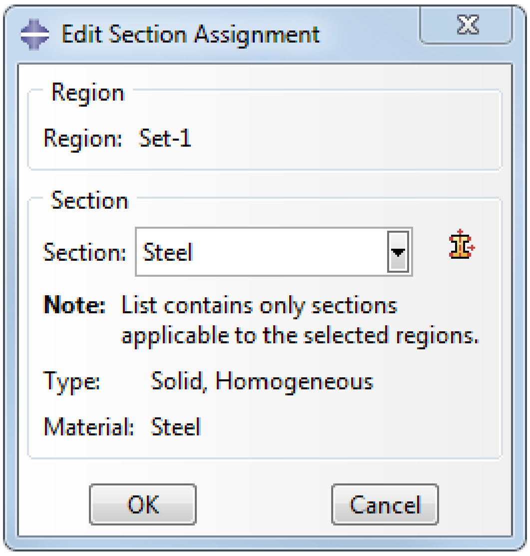
and click **Continue**.

The **Edit Section** dialog box will open. Select there **Material = Steel** and click **OK**.



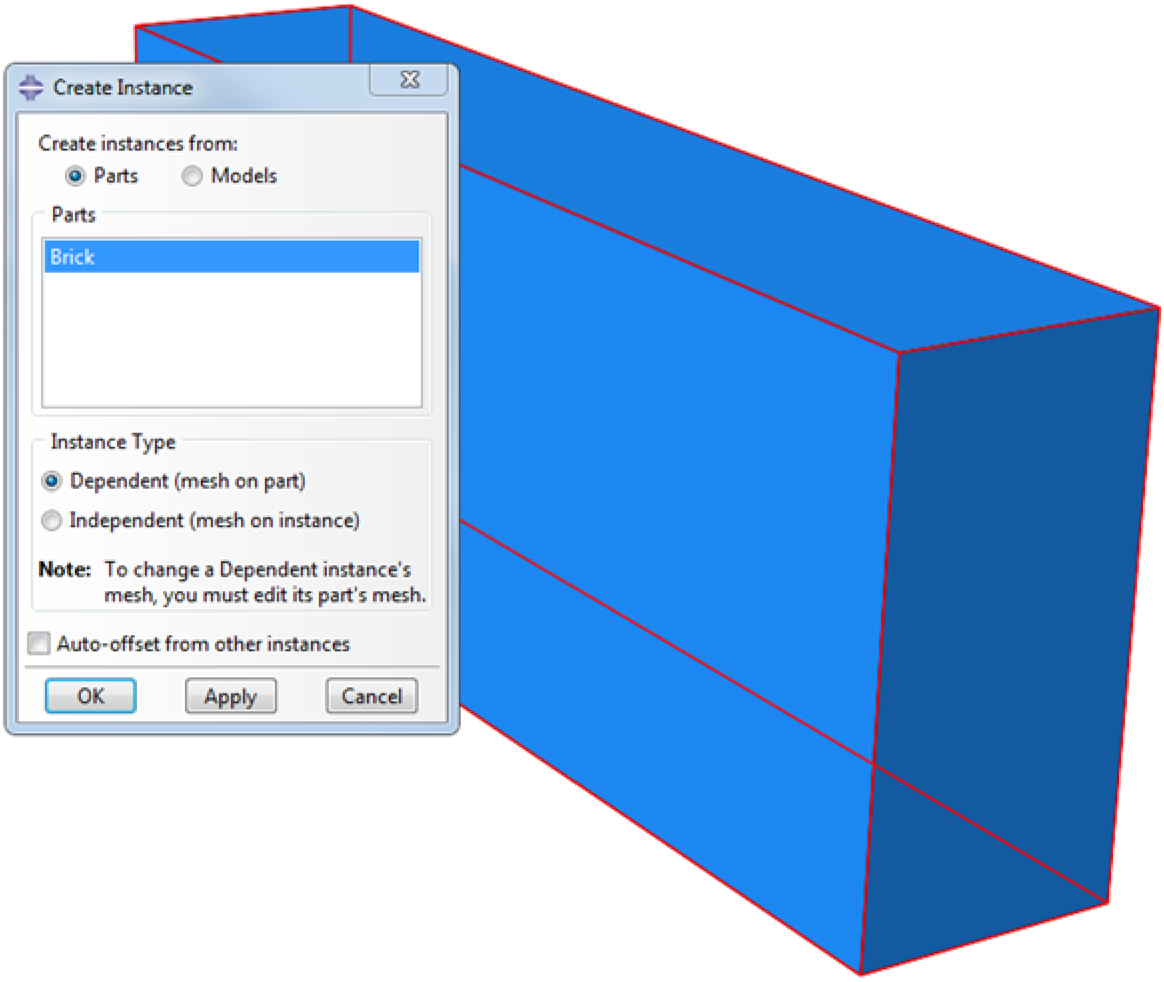
Creating and editing a section

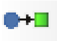
7. Click  (**Assign Section**) and in the dialog box **Edit Section Assignment** assign the just created section to the part, which you have created. Select the whole brick (it will be highlighted by red) and click **Done**. In the dialog box, which opens, click **OK**.



Assigning a section to the part Brick

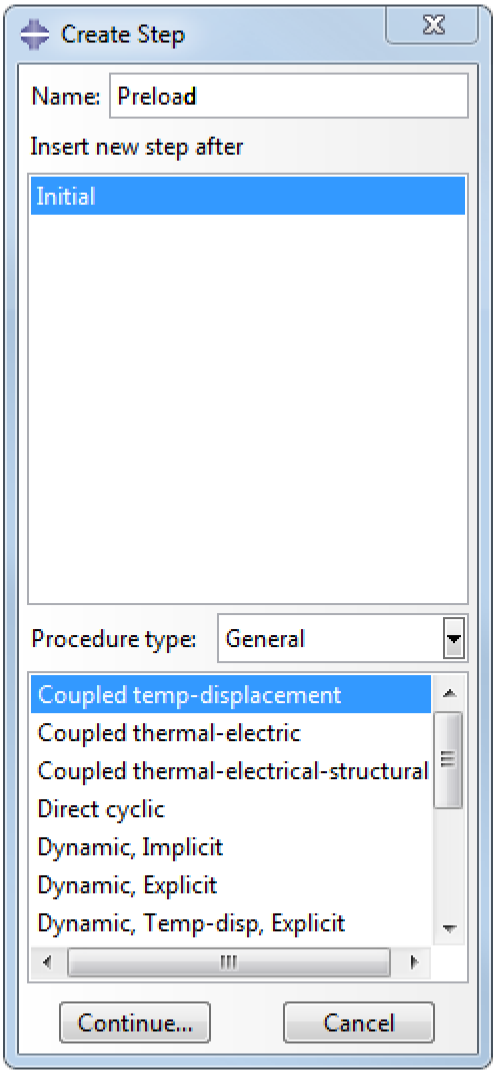
8. Creating an assembly and setting the analysis. Navigate to the module **Assembly**. To create an instance, click  (**Create Instance**). A window will open with a list of created parts. Select the part **Brick** and specify **Instance type = Dependent**. The just created part will appear in the graphical work window. Click **OK**.



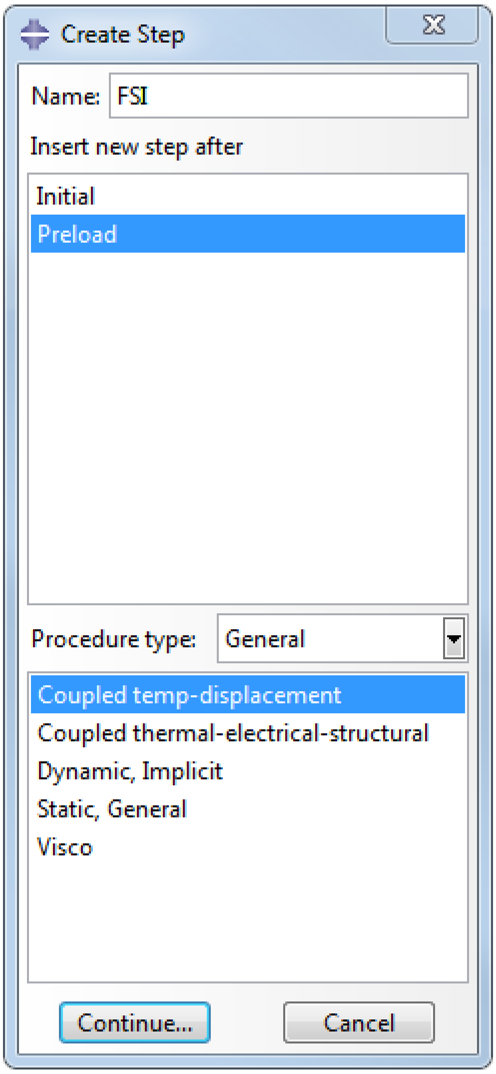
9. Specify the procedure of analysis for the preliminary computation. The numerical simulation will consist of two steps, the first step is used for the preliminary computation of heating the brick. Navigate to the module **Step**, click  (**Create Step**). In the dialog box **Create Step** specify:

Name = Preload

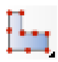
Procedure Type = General
From the list below select **Coupled temp-displacement** and click **Continue**.

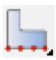


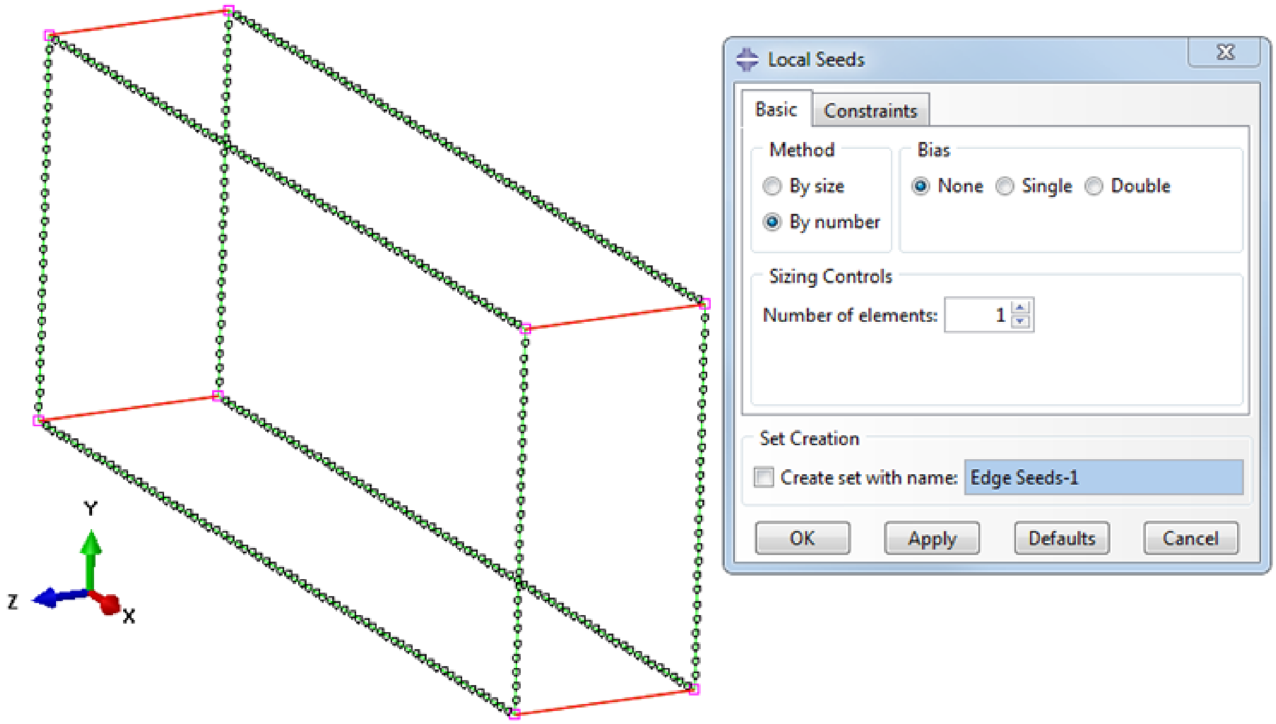
10. The **Edit Step** dialog box will open where you have to specify parameters of the step, which will be used in the analysis. In the **Basic** tab specify:
    Response = Steady-state
    Time Period = 0.1
    NIgeom = Off
    Don't change the default settings in the **Incrementation** tab.
11. Create the second step of the analysis. At this step the FSI computation will be done. In the dialog box **Create Step** specify:
    Name = FSI
    Procedure Type = General
    From the list below select **Coupled temp-displacement** and click **Continue**.



12. The **Edit Step** dialog box will open. Specify there in the **Basic** tab:
    Response = Steady-state
    Time Period = 20
    Nlgeom = Off
    In the **Incrementation** tab specify:
    Maximum numbers of increments = 100000 (maximal number of increments)
    Increment size > Initial = 0.01 (the initial increment)
    Increment size > Minimum = 1E-10 (the minimal increment)
    Maximum increment size > Specify = 10 (the maximal increment)
    Click **OK**.

13. Navigate to the module **Mesh Object: Part**. Apply the tool  (**Seed part Instance**) to split the brick into elements. Specify **Approximate global size = 0.0015** and click **OK**.

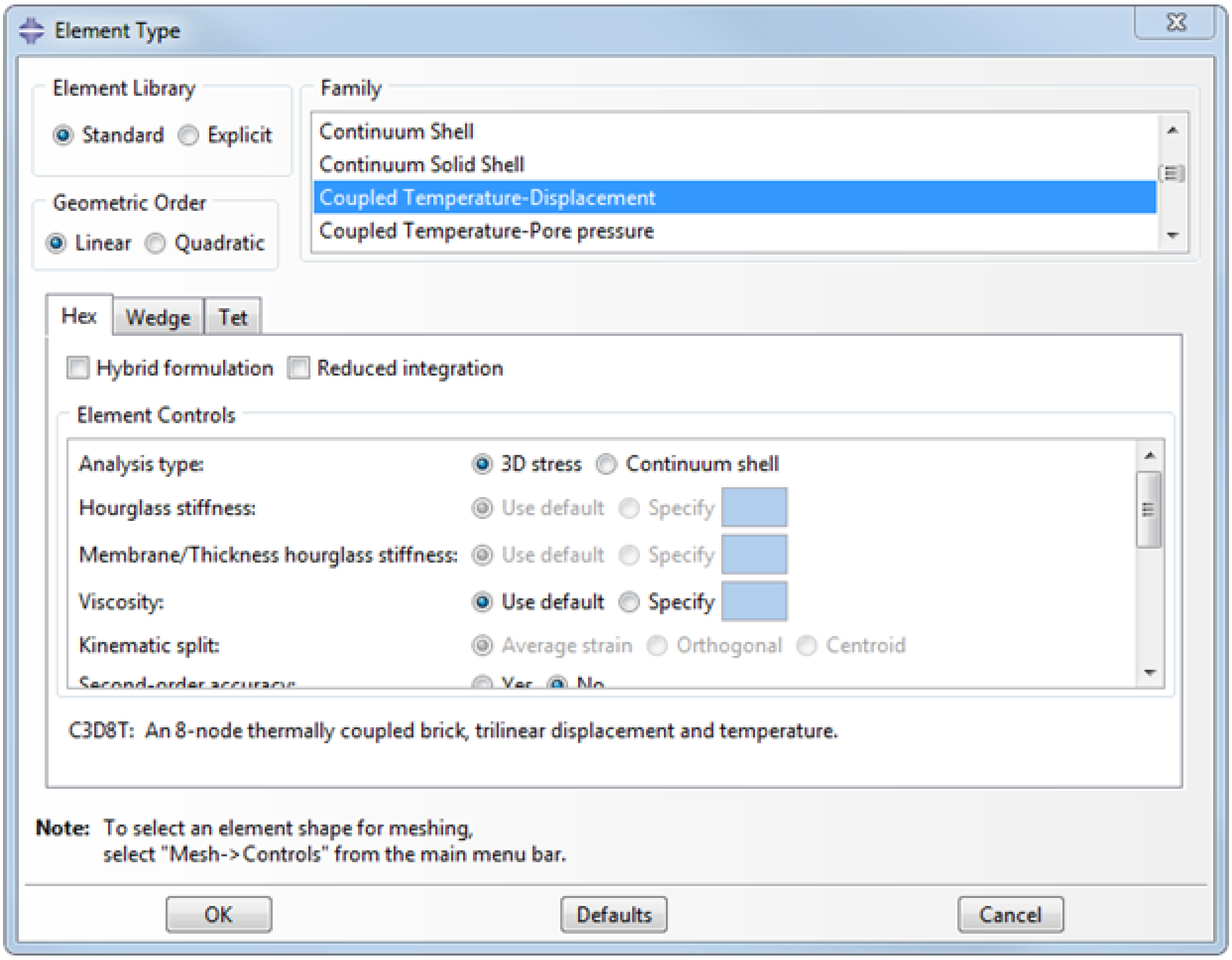
    Apply the tool  (**Seed edges**) and specify 1 element on each face along the axis Z.



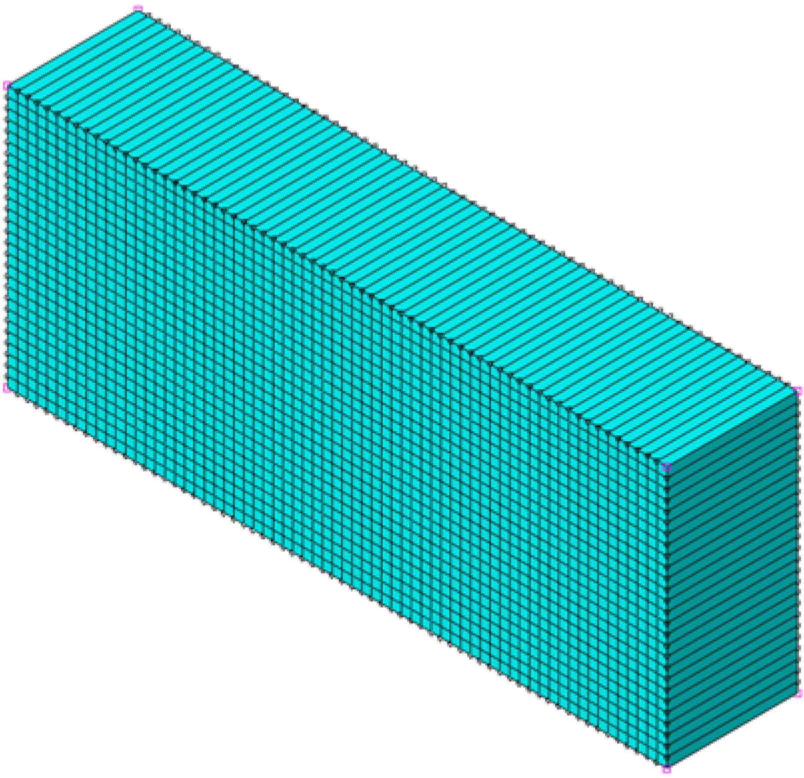
Splitting the geometry

For this type of analysis you have to select the appropriate type of elements (**C3B8T**). Click  (**Assign Element Type**) and specify the following settings:

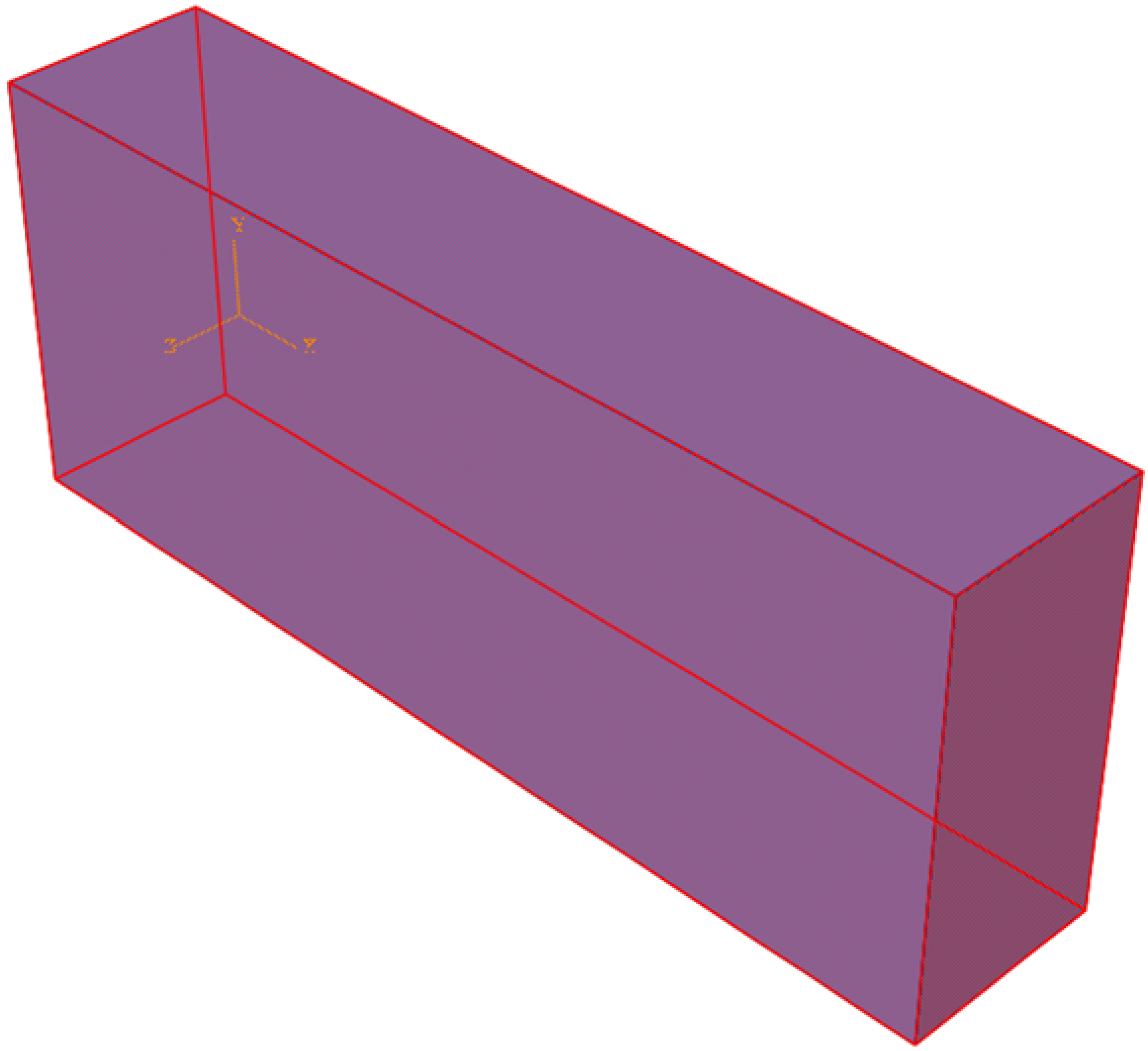
Element Library = Standard
Geometric Order = Linear
Family = Coupled Temperature-Displacement



13. To start the grid generation, click (**Mesh Part Instance**) and then click **Yes**.



6.4.1.2 Specifying an interface surface in Abaqus



The next important step of creation the model for joint computation is specifying an ***interface surface*** between *Abaqus* and *FlowVision*.

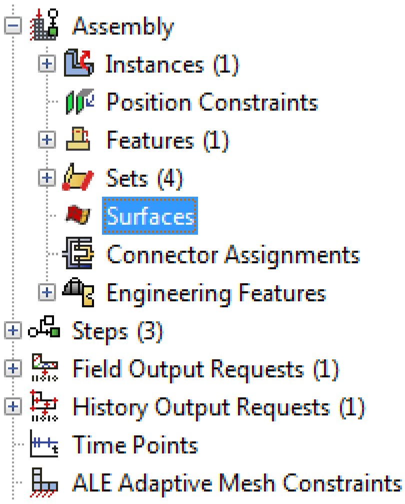
Abaqus simulates deformations and thermal flows within the brick and exports temperature from nodes on the brick's surface to *FlowVision*.

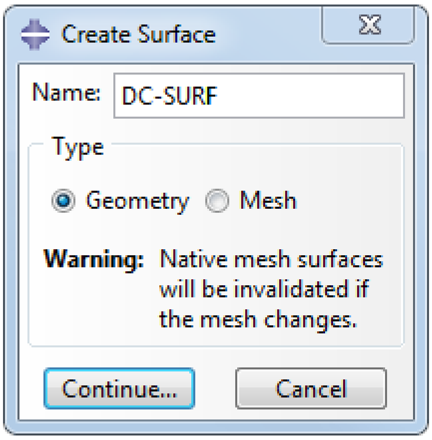
Follow these steps:

In the model tree double-click **Assembly > Surfaces** (or select and use the **Create** command). The **Create Surface** dialog box will open; specify there:

Name = DC-SURF (name of the interface surface)

Type = Geometry



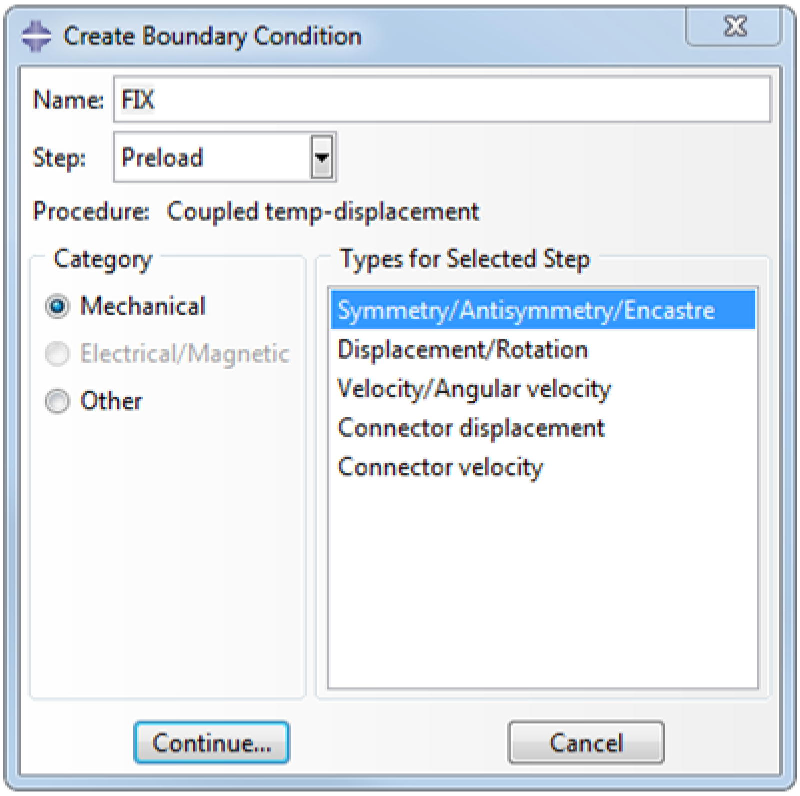


Click **Continue** and select the whole brick. Click **Done**.

6.4.1.3 Specifying boundary conditions and loads in Abaqus

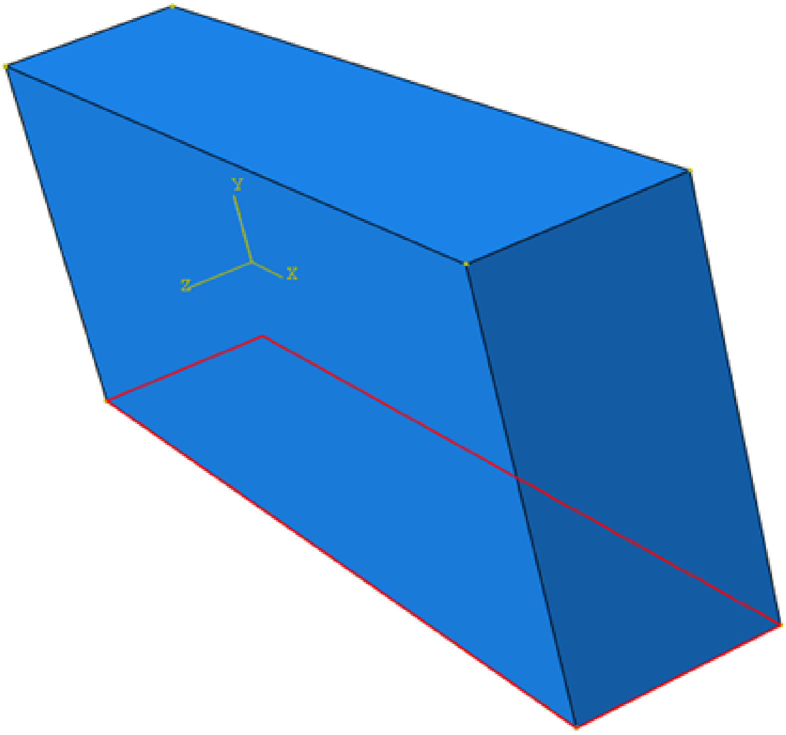
Follow these steps:

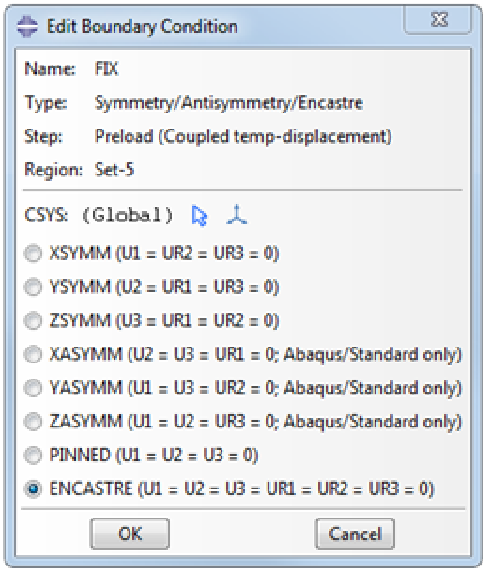
1. Navigate to the module **Load**. Select  (**Create Boundary condition**) and create a boundary condition on the bottom surface of the brick with the following parameters:
Name = FIX
Step = Preload
Category = Mechanical
Types for Selected Step = Symmetry/Antisymmetry/Encastre



Click **Continue**.

2. In the graphical work window select the bottom surface of the brick and click **Done**. In the **Edit Boundary Condition** dialog box, which opens, select for the brick's face the boundary condition type **ENCASTRE (U1=U2=U3=UR1=UR2=UR3=0)** that prohibits movement of this face for any degrees of freedom:

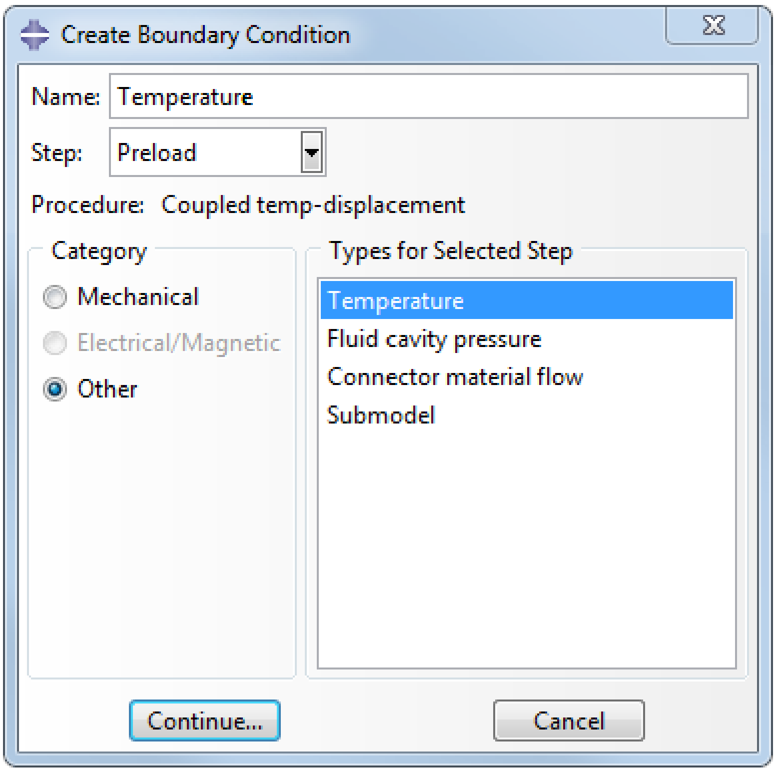




3. Create a new boundary condition (on the same surface of the brick) with the following parameters:

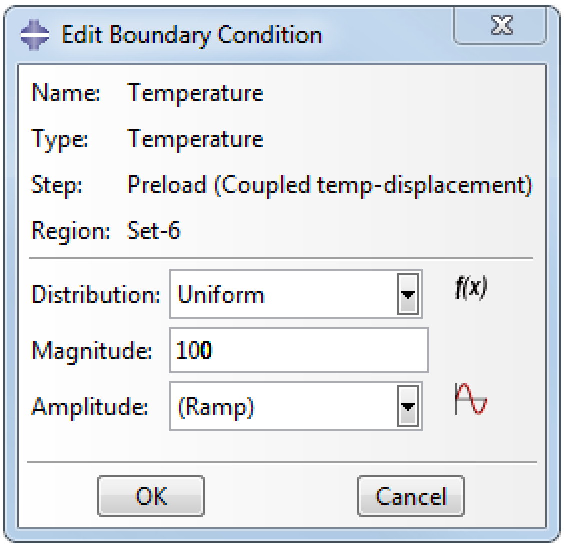
Name = Temperature
Step = Preload
Category = Other
Types for Selected Step = Temperature

Click **Continue**.



4. In the graphical work window select the bottom surface of the brick and click **Done**.

In the **Edit Boundary Condition** dialog box, which opens, specify **Magnitude=100**.



6.4.1.4 Generating an inp-file

To generate an `inp`-file, follow these steps:

1. Navigate to the module **Job** and in the window **Job Manager** () create a new analysis.
   Click **Create**; after this the **Create job** dialog box will open; specify there the name **BrickABQ** for the new analysis and click **Continue**.
2. In the **Job Manager** window select the line of the just created analysis and click **Write Input** to create an input file, geometry of which will be imported to *FlowVision*. Also this input file will be required to make the computation.
3. Save and close the project.

6.4.1.5 Modifying the inp-file of the Abaqus' project

The ready file **`BrickABQ.inp`**, which is included into the delivery, has already been modified. If you use the file, you can skip this section, but we recommend to open and browse contents of this file using any text editor.

If you wish to create the **`BrickABQ.inp`** file by your own, follow these steps:

- Open the **`inp`**-file of the *Abaqus*' project in a text editor.
- Add the following lines into the module **STEP** for the step **`**Step: FSI`** the following lines before the line **`*End Step`**:

```
*CO-SIMULATION, PROGRAM=DIRECT, NAME=FlowVision, CONTROLS=FSI
*CO-SIMULATION REGION, IMPORT
ASSEMBLY_DC-SURF, CFL
*CO-SIMULATION REGION, EXPORT
ASSEMBLY_DC-SURF, NT
*CO-SIMULATION CONTROLS, NAME=FSI, TIME INCREMENTATION=SUBCYCLE, TIME MARKS=YES
```

  These lines are used to identify the coupling analysis with another program.

Important notice: you have to add **`ASSEMBLY_`** before the interface region's name in lines of the co-simulation. After the modification the **`inp`** file have not to contain empty lines.

See also: More details about these settings (**`*CO-SIMULATION`**, **`*CO-SIMULATION CONTROLS`**, **`*CO-SIMULATION REGION`**) you can found in the *Abaqus Keywords Reference Manual.*

6.4.2 Preparing the project in FlowVision

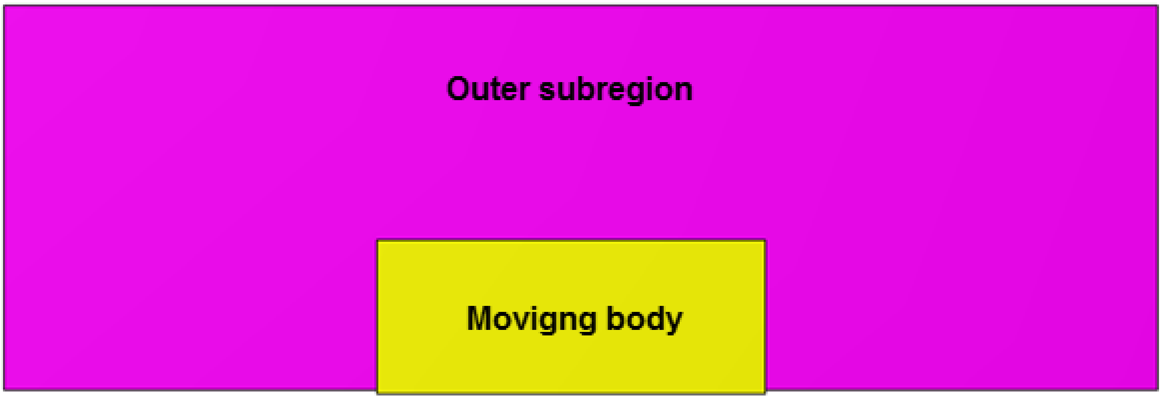
FlowVision simulates motion and heating the cold air in the channel.

Requirements to the *FlowVision's* project:

- The geometry of the deformable body has to be loaded from *Abaqus* into *FlowVision*, and this geometry must completely comply to the deformable geometry in the *Abaqus* project.
- The geometry of the deformed body, loaded into *FlowVision*, must comply with the requirements to the geometry in *FlowVision* (see *User's guide*).

6.4.2.1 Geometry

Geometry for this exercise is created in the **Geometry** tab in **Pre-Postprocessor**.



Open **Pre-Postprocessor**, apply the **File > Create** command from the main menu, and in the **New project** dialog box, which opens, select **Create empty project**.

In the **Geometry** tab apply the **Create** command from the context menu of the **Initial geom. models** folder to create a **Box** object.

In the **Properties** window of the new **Box** specify:

Object > Location

Reference point

X	**= 0.075**	[m]
Y	**= 0**	[m]
Z	**= 0**	[m]

Axis X

X **= 1**
Y **= 0**
Z **= 0**

Axis Y

X **= 0**
Y **= 1**
Z **= 0**

Object > Size

X	**= 0.45**	[m]
Y	**= 0.018**	[m]
Z	**= 0.1**	[m]

From the context menu of **Box #0** select the command **Create consistent mesh** and specify **Approximation parameters** for the new consistent mesh:

Size 1 = 1
Size 2 = 1
Size 3 = 1

From the context menu of the new consistent **mesh** select the command **Use in SubRegion Composer**. The **Box #0 - mesh** element will appear in the subfolder **SubRegion Composer > Objects**.

From the context menu of the element **SubRegion Composer > Composed subregions** select **Compose**. The **Cone/cylinder #0 - mesh** element will appear in the subfolder **SubRegion Composer > Composed subregions**.

After the composing, the object **Box #0 - mesh** can be used as the **Region's** geometry in the **Preprocessor** tab. Rename the object **Box #0 - mesh** as **Subregion #0**. From the context menu of the folder **Composed subregions** select **Use as Region main geometry**.

6.4.2.2 Physical model

In this exercise, the **Substance** is air. The **Substance** will be loaded from the standard **Substance database**.

- Open the context menu of the folder **Substances** and, by the command **Create**, create **Substance #0**.
- From the context menu of **Substance #0** select the command **Load from SD > Standard** and load the substance **Air** in its gas phase.

In the folder **Phases**:

- Create a continuous **Phase #0**.
- In **Phase #0** add **Substance #0** into the subfolder **Phase #0 > Substances**.
- In properties of the folder **Phase #0 > Physical processes** specify:

Heat transfer	**= Heat transfer via h**
Motion	**= Navier-Stokes model**
Turbulence	**= KES**

In the folder **Models**:

- Create **Model #0**.
- Add **Phase #0** into subfolder **Model #0 > Phases**.
- Specify motion of the fluid along the axis X with velocity 2 [m/s]. To do so, in properties of the element **Models > Model #0 > Init. data > Init. data #0 > Velocity (Phase #0)** specify:

X **= 2** [m s^{-1}]

In properties of **SubRegion #0** specify **Model = Model #0**.

6.4.2.3 Creating the Imported object and the Moving body

The brick is inserted into the project as an imported geometry **Object**, on which the **Moving body** modifier is set.

Follow these steps:

In the folder **Objects**:

- From the context menu of the folder **Objects** select the command **Create** and in the **Create new object** dialog box, which opens, select **Object type = Imported object**.
- From the dialog box, which opens, select the file `BrickABQ.inp`. **Imported object #0** will appear in the folder **Objects**.

Create a **Moving body** modifier on **Imported object #0**:

- From the context menu of the folder **Subregions > SubRegion #0 > Modifiers** select the command **Create** and in the **Create new modifier** dialog box, which opens, select:

  Modifier type = Moving body
  Objects = Imported object #0

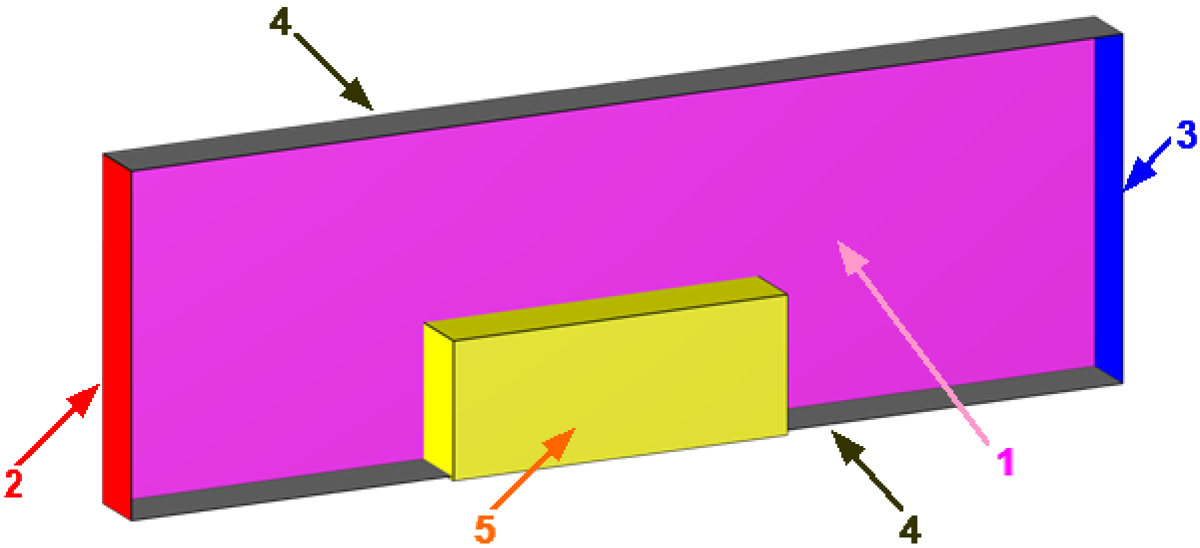
  The **Moving body #0** modifier will appear in the folder **Subregions > SubRegion #0 > Modifiers**.
- In properties of the just created modifier **Moving body #0** specify its initial position:

  Initial position
  Reference point
  X **= -0.04** [m]

	Y	**= 0.01**	[m]
	Z	**= -0.041**	[m]
Axis Y			
	X	**= 0**	
	Y	**= 0**	
	Z	**= 1**	

- Click **Apply** and then place **Moving body #0** to its initial position (click the icon **Operations >**).

6.4.2.4 Boundary conditions



Specify boundary conditions and assign them according to the illustration above.

Boundary 1

Name	**= Symmetry**	
Type	**= Symmetry**	
Color	**= Fuchsia**	

Boundary 2

Name	**= Inlet**	
Type	**= Inlet/Outlet**	
Variables		
Temperature (Phase #0)	**= Temperature**	
Value	**= 0**	[°C]
Velocity (Phase #0)	**= Fixed velocity**	
Value > X	**= 2**	[m/s]
TurbEnergy (Phase #0)	**= Pulsations**	
Value	**= 0**	
TurbDissipation (Phase #0)	**= Turbulent scale**	
Value	**= 0**	[m]
Color	= **Red**	

Boundary 3

Name	**= Outlet**	
Type	**= Free Outlet**	
Variables		
Temperature (Phase #0)	**= Zero gradient**	
Velocity (Phase #0)	**= Total pressure**	
Value	**= 0**	[Pa]
TurbEnergy (Phase #0)	**= Zero gradient**	
TurbDissipation (Phase #0)	**= Zero gradient**	
Color	= **Blue**	

Boundary 4

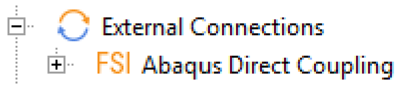
Name	**= Wall**
Type	**= Wall**
Variables	
Temperature (Phase #0)	**= Zero gradient**
Velocity (Phase #0)	**= Logarithm law**
TurbEnergy (Phase #0)	**= Value in cell near wall**
TurbDissipation (Phase #0)	**= Value in cell near wall**
Color	= **Gray**

Boundary 5

Name	**= Brick**
Type	**= Wall**
Variables	
Temperature (Phase #0)	**= External conjugate**
Velocity (Phase #0)	**= Logarithm law**
TurbEnergy (Phase #0)	**= Value in cell near wall**
TurbDissipation (Phase #0)	**= Value in cell near wall**
Color	= **Yellow**

6.4.2.5 Parameters of co-simulation

- From the context menu of the **External Connections** folder select the **Create** command.
- In the **Create new object** dialog box, which opens, select **Object type = Abaqus Direct Coupling**.
- A dialog box for access to files will open; select there the `BrickABQ.inp` file.
- An **Abaqus Direct Coupling** child element will appear in the **External Connections** folder:



The **Abaqus Direct Coupling** element, in its turn, has a child element **ASSEMBLY_DC-SURF**.

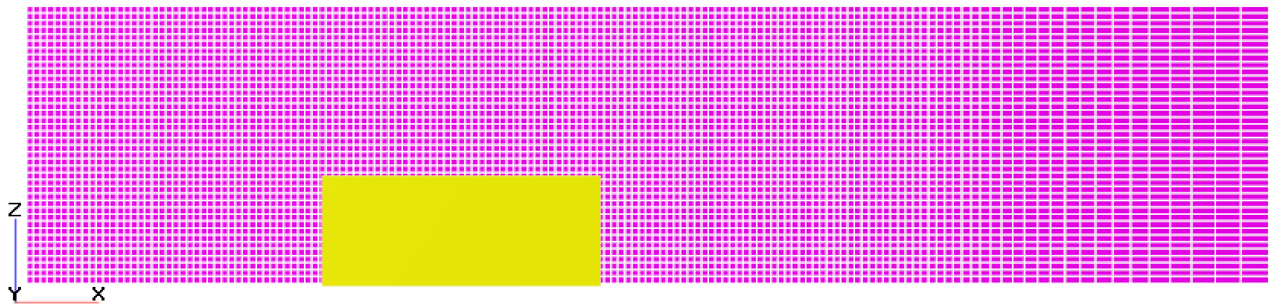
- In properties of the folder **External Connections** specify:
  Exchange step > Method = FlowVision
  Exchange step > Default time step = 0.01
  Exchange step > Coef. for time step = 1
  Old offset = Yes
  AutoSave = Yes
- In properties of the element **Abaqus Direct Coupling** specify:

Activation = Yes
ABAQUS > Run ABAQUS = Yes
ABAQUS > MPM Agent > Address = host name or IP address of the computer, on which **MPM Agent** will run that will start *Abaqus*.
ABAQUS > MPM Agent > Port = the default port, which is used by **MPM Agent** of the current version
ABAQUS > IP Source = IP of MPM Agent
ABAQUS > Port = 5555
ABAQUS > Timeout [s] = 100
Loads relaxation > Scale factor = 1
Loads relaxation > Start in steps = 0
Loads relaxation > End in steps = 0
Loads relaxation > Initial coefficient = 1
Loads relaxation > Final coefficient = 1
Heat relaxation > Scale factor = 1
Heat relaxation > Start in steps = 0
Heat relaxation > End in steps = 20
Heat relaxation > Initial coefficient = 0
Heat relaxation > Final coefficient = 1

- In properties of the element **ASSEMBLY_DC-SURF** specify **Moving body = Moving body #0**.

Don't change other settings.

6.4.2.6 Initial grid



In the **Properties** window of the **Initial grid**, click the button to open the **Initial grid editor**.
Specify in the **Initial grid editor**:

for axis OX

Grid parameters

h_max	**= 0.01**	[m]
h_min	**= 0.0025**	[m]

Insert a reference line with coordinate:

x = 0.15	[m]

Specify **Reference line parameters** for the reference line with coordinate **x=-0.15**:

h	**= 0.0025**	[m]
kh+	**= 1**	

Specify **Reference line parameters** for the reference line with coordinate **x=0.15**:

h	**= 0.0025**	[m]
kh-	**= 1**	
kh+	**= 1**	

Specify **Reference line parameters** for the reference line with coordinate **x=0.3**:

h	**= 0.01**	[m]

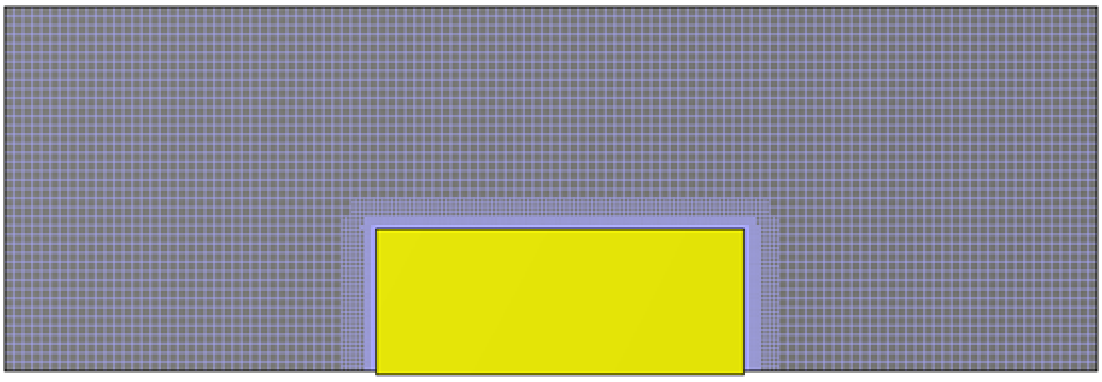
Specify in properties of the **Initial grid**:

Grid structure = 2D
Plane = XZ
nZ = 40

In the **Properties** window of the **Initial grid** click **Apply**.

6.4.2.7 Adaptation of the grid

Creating the Adaptation



We need to apply the adaptation of the computational grid gradually; refinement of the grid will increase in the boundary layer during the computation.

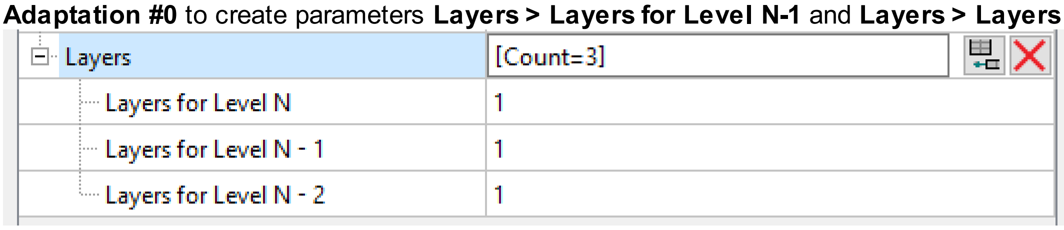
This is achieved by gradual adaptation of the grid from the 1st to the 3rd level within 300 steps; each new level of adaptation turns on after next 100 steps of computation.

Follow these steps to specify the adaptation:

- Create the element **Computational grid > Adaptation > Adaptation #0**.
- Add the boundary condition **Brick** into the folder **Computational grid > Adaptation > Adaptation #0 > Objects**.
- In properties of **Adaptation #0** specify the **Max level N** parameter as a table of dependency on the **Current step number**. Select **Table f(x)** on the right from the **Max level N** field and then click the button to open the **Table editor**. In the **Table editor**, in the argument column (**Arg#0**), click button and then select (**Bind to variable**). Then, in the **Variable selection** dialog box, which opens, select **Integral > Internal characteristics > Current step number** and specify the tabulated function:

x	f(x)
0	0
100	1
200	2
300	3

- In our case the adaptation keeps thickness of the adapted layer. Adaptation layers will also be specified by tables. When a new adaptation layer is enabled, boundary of the 1st level of adaptation don't change and all new adaptations will be created within the 1st layer, spending the required number of cells. The argument variable, which will be used for tabulated values of numbers of adaptation layers, will also be **Current step number**. Click twice the button (**Append item to the array**) in the **Layers** line in properties of **Adaptation #0** to create parameters **Layers > Layers for Level N-1** and **Layers > Layers for Level N-2**:



- Specify the tabulated dependency of **Layers for Level N** on **Current step number**:

x	f(x)
0	0
100	7
200	6
300	4

- Specify the tabulated dependency of **Layers for Level N-1** on **Current step number**:

x	f(x)
0	0
200	4

- Specify the tabulated dependency of **Layers for Level N-2** on **Current step number**:

x	f(x)
0	0
300	4

6.4.2.8 Parameters of calculation in FlowVision

This FSI simulation assumes only heat exchange in the interaction between *FlowVision* and *Abaqus*.

Because of there is no import of coordinates into *FlowVision*, the time step will be defined only by convective and diffusive CFLs. The surface CFL is not taken into account because of there are no replacing or motion of the **Moving body**.

In the **Solver** tab, in properties of the **Time step** element, specify:

Method = **Via CFL number**
Convective CFL = **100**
Surface CFL = **1E+20**
Diffusive CFL = **100**

6.4.2.9 Visualization

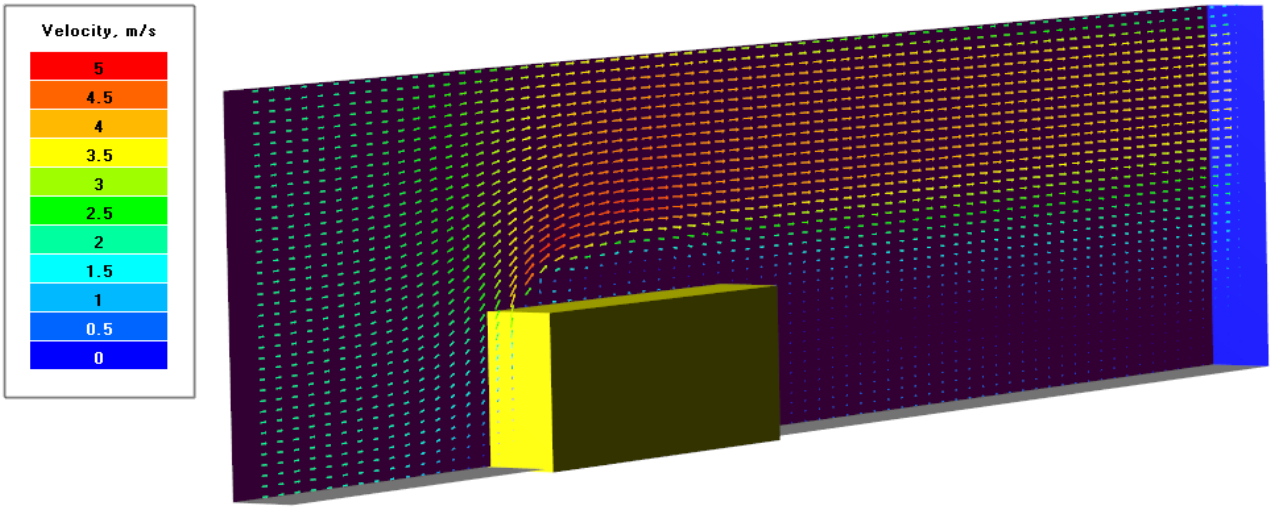
Visualization is shown for the step number near 650.

Open the **Postprocessor** tab.

In properties of the object **Objects > Plane #0** specify:

Object
Normal
X = **0**
Y = **1**
Z = **0**

Velocity distribution



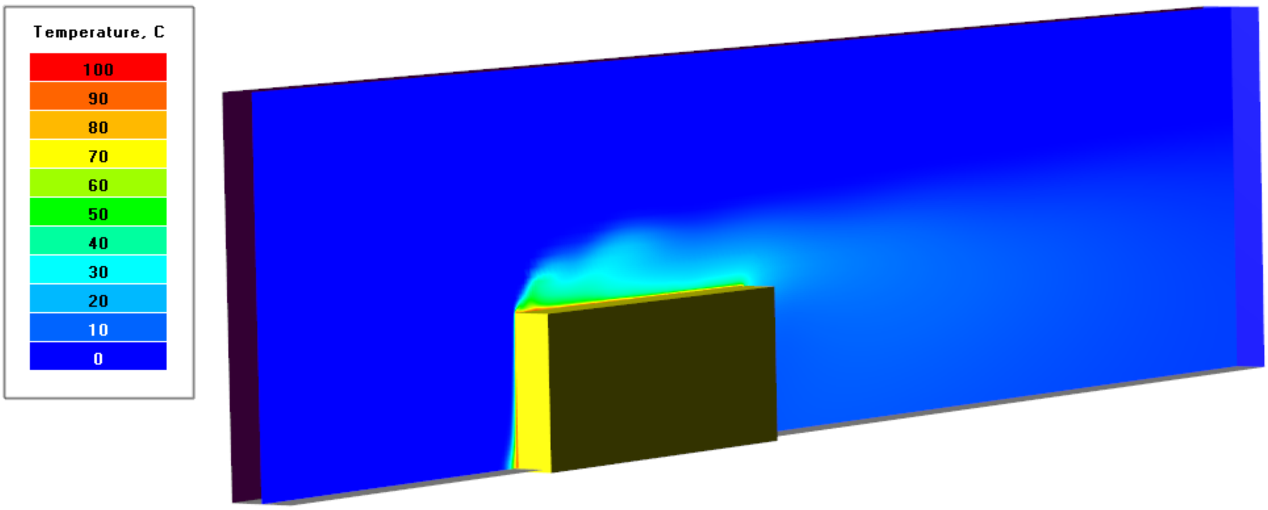
From the context menu of the folder **Layers** select **Create**, and in the dialog box **Create new layer** specify:

Layer type = Vectors
Objects = Plane #0

In properties of the new **Layer** specify:

Name = Velocity, m/s
Variable > Variable = Velocity
On regular grid = Yes
Grid > Size 1 = 70
Grid > Size 2 = 40
Scaling > Mode = Manual
Scaling > Maximum = 5
Scaling > Reference length = 0.005
Coloring > Variable > Variable = Velocity
Coloring > Value range > Mode = Manual
Coloring > Value range > Max = 5
Coloring > Value range > Min = 0
Coloring > Palette > Appearance > Enabled = Yes
Coloring > Palette > Appearance > Title = Yes
Coloring > Palette > Appearance > Style = Style 1
Coloring > Palette > Appearance > Color = Black

Temperature distribution on Plane #0



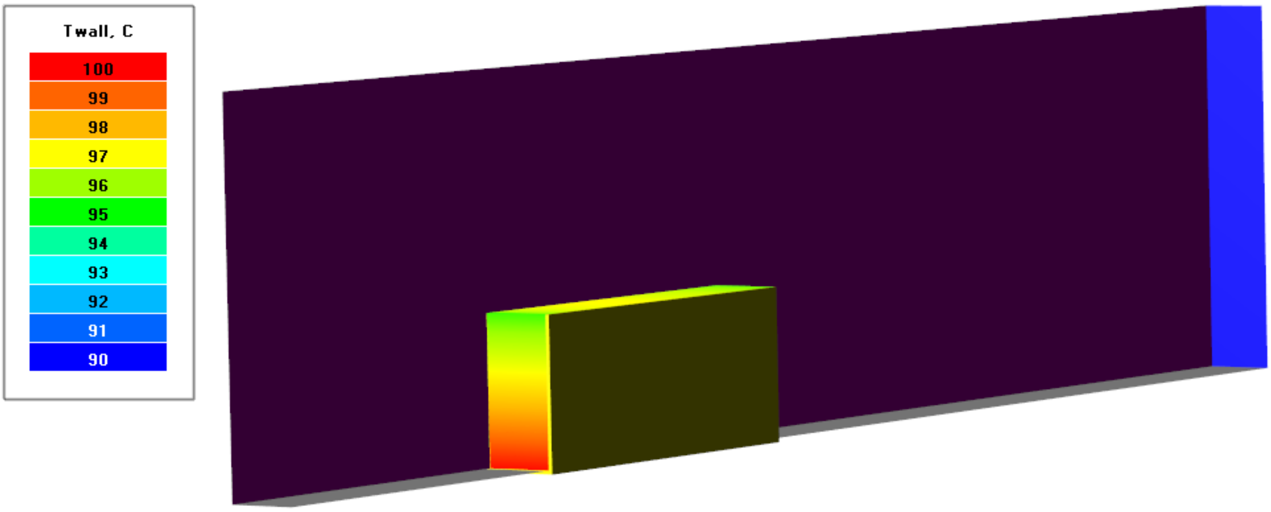
From the context menu of the folder **Layers** select **Create**, and in the dialog box **Create new layer** specify:

Layer type = Color contours
Objects = Plane #0

In properties of the new **Layer** specify:

Name = Temperature, C
Variable > Variable = Temperature
Value range > Mode = Manual
Value range > Max = 100
Value range > Min = 0
Palette > Appearance > Enabled = Yes
Palette > Appearance > Title = Yes
Palette > Appearance > Style = Style 1
Palette > Appearance > Color = Black

Temperature distribution on the Imported Object



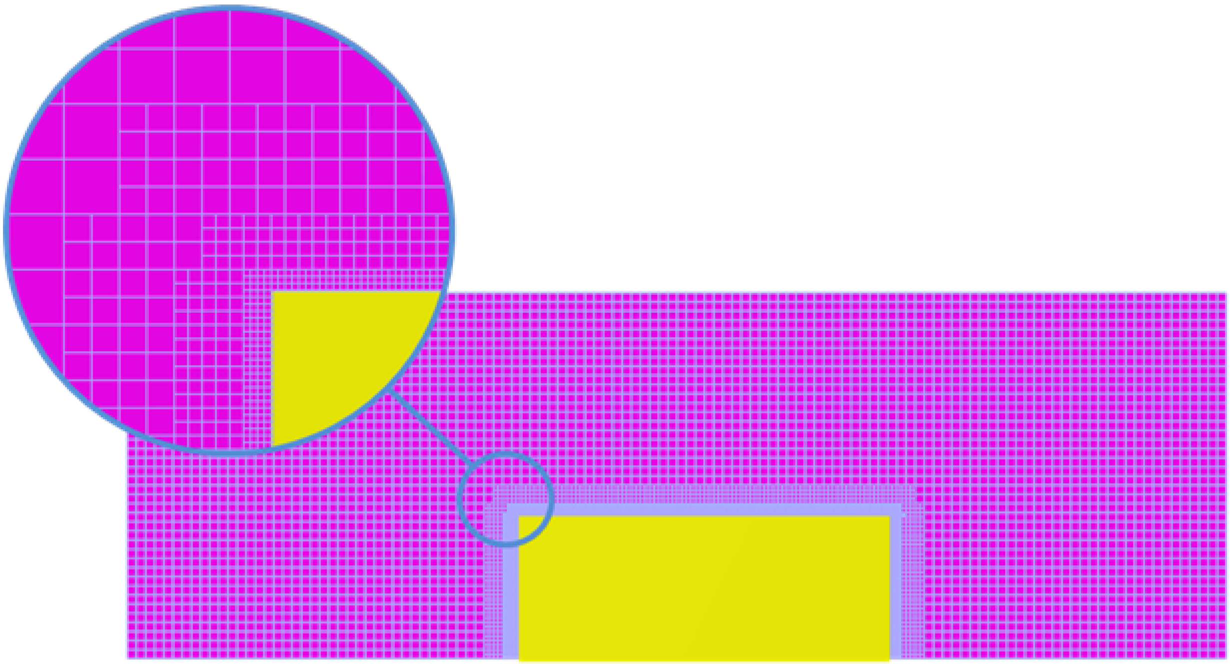
From the context menu of the folder **Layers** select **Create**, and in the dialog box **Create new layer** specify:

Layer type = Color contours
Objects = Imported object #0

In properties of the new **Layer** specify:

Name = Twall, C
Variable > Variable = Temperature
Value range > Mode = Manual
Value range > Max = 100
Value range > Min = 90
Palette > Appearance > Enabled = Yes
Palette > Appearance > Title = Yes
Palette > Appearance > Style = Style 1
Palette > Appearance > Color = Black

Cross-section of the computational grid



From the context menu of the folder **Layers** select **Create**, and in the dialog box **Create new layer** specify:

Layer type = Computational grid section
Objects = Plane #0

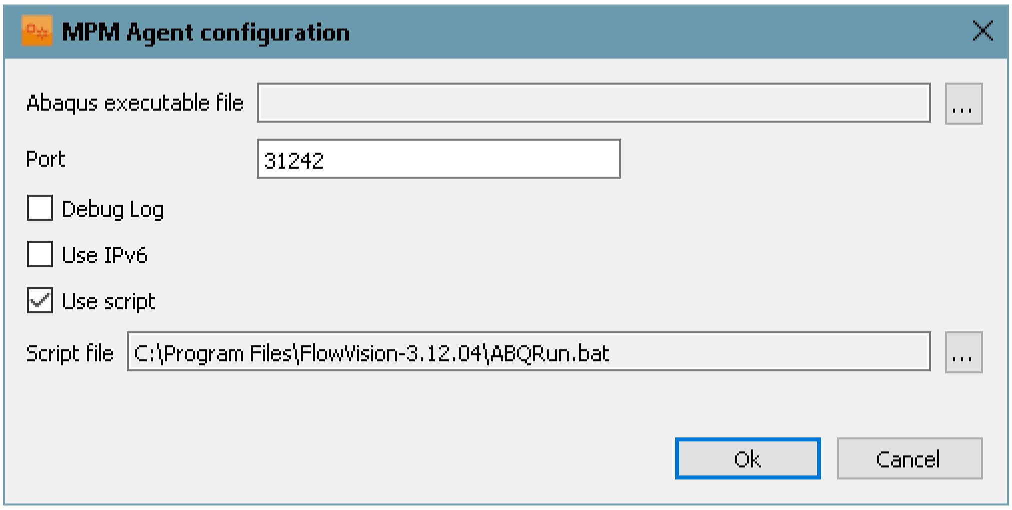
In properties of the new **Layer** specify:

Name = Computational grid section
Appearance > Mode = Lines

6.4.3 Starting and stopping the computation

Before starting the computation you have to run **MPM-Agent**.

MPM-Agent has to receive the path to the executable file of *Abaqus*. If the path has not been provided, you can specify it in the *FlowVision*'s module **Configurator**. To do so, click in **Configurator** the **View** button on the right near the **MPM Agent** field in the **Configuration files** pane in the **Configuration/Logs** tab and specify the data in the **MPM Agent configuration** dialog box, which opens:



The project's computation is started from **Pre-Postprocessor**, see description of a standard starting in the section [Detailed description of a simplest model > Laminar flow in a tube > Starting the computation](#).

Stopping after 20 seconds of the simulated time is set in *Abaqus*. Also you can stop the project using the  (**Stop computation**) button, and then resume the computation after editing the project.

6.5 Joint simulating flow of liquid and heat exchange by FlowVision and APM WinMachine

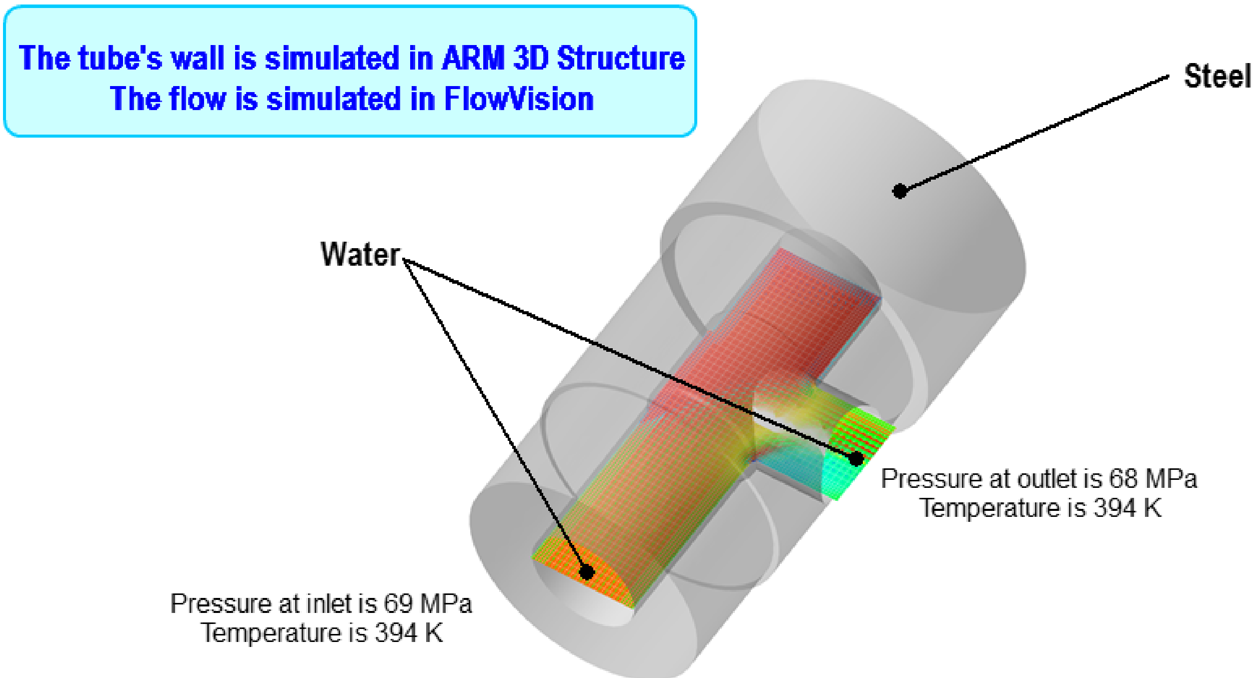
This exercise demonstrates abilities of joint simulating made by *FlowVision* and a third-party FEA software *APM WinMachine*.

> *APM WinMachine* is a CAE-software for automated analysis and design of mechanical equipment, developed with the latest advances in computational mathematics, calculus and software coding, and also theoretical and trial engineering solution.

In this problem setting we examine flow of liquid in steel tube under high pressure. The high pressure is applied to walls and causes deformations of flow passage of the tube and increases the liquid flow rate.

Besides simulating the strain-stress state, heat transfer is also simulated in this problem. The temperature drop between the outer part of the tube and the flow passage of the tube causes temperature strains of the assembly.

The applied approach allows you to evaluate deformations of the assembly along with liquid flow rate through the tube's flow passage calculated with the deformations taken into account.



The project is created using the following software products:

- *APM Studio 19*: creation geometry and mathematical FEA model.
- *APM Structure3D 19*: creation the exchange surface, starting the joint computation in *APM WinMachine* and calculations of the FEA model:
    - calculation of the strain-stress state
    - simulating the heat transfer to outside environment.
- *FlowVision*: preparing a project, starting the joint computation in *FlowVision*, and computation of the liquid's flow in the flow passage.

Files

Geometry model that is to be imported by *APM Studio*	`PipeAPM.stp`
Geometry model of the **Imported object** in *FlowVision*	`ClosedSurf.mesh`
Project in *FlowVision*	`APM_HeatExchange`

6.5.1 Preparing a project in APM Studio

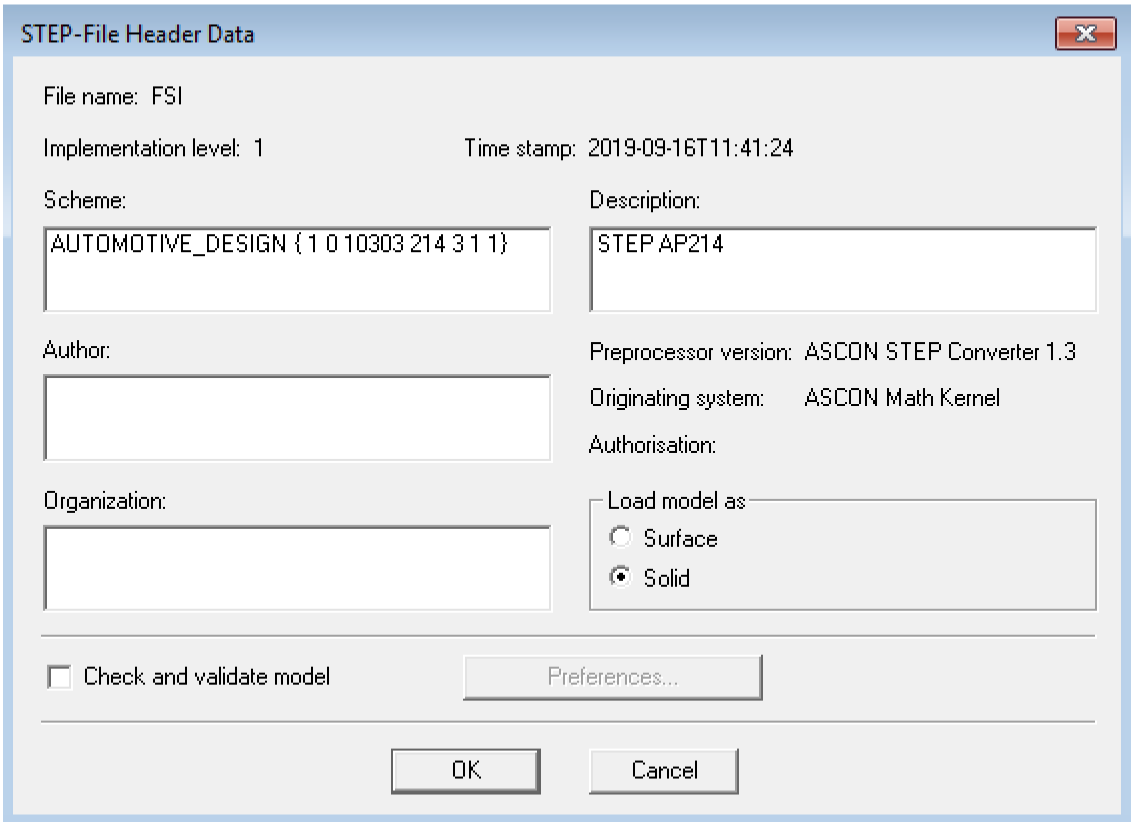
Open *APM Studio* and follow steps described in subsections below:

- Importing the geometry model to APM Studio
- Boundary and initial conditions for the thermal calculation
- Specifying restraints for strength calculation
- Specifying properties of material of the part
- Creating and saving the finite-element mesh

6.5.1.1 Importing the geometry model to APM Studio

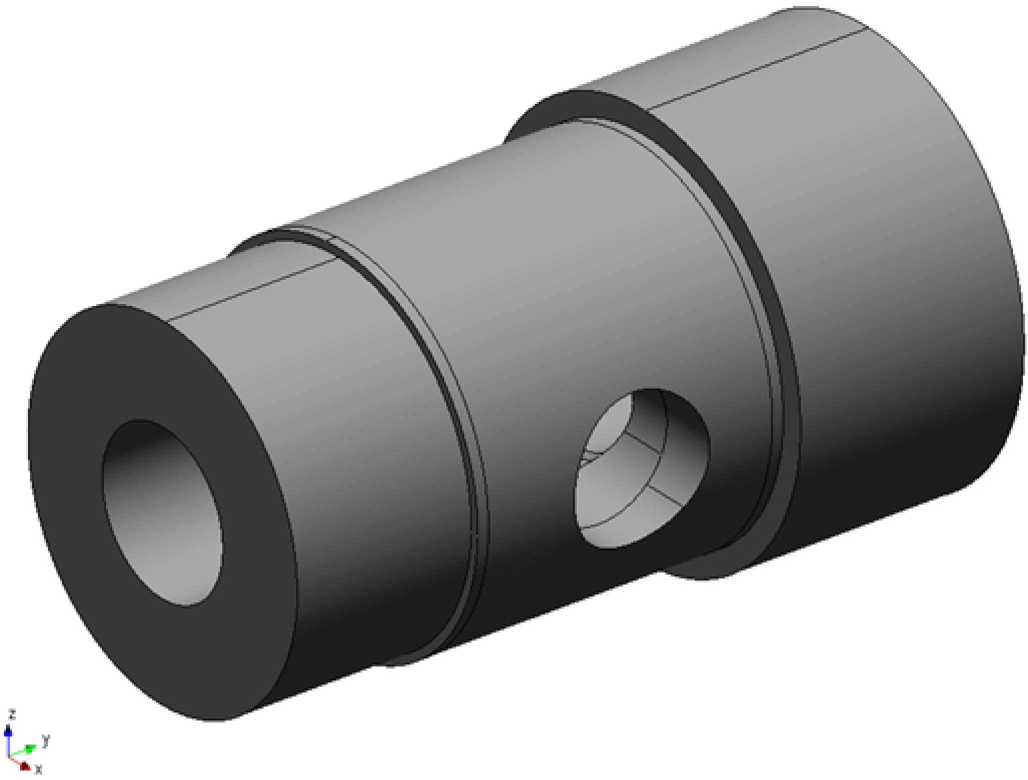
In the *APM Studio* software apply the command **File > Import** and select the file **`PipeAPM.stp`**, which contains the geometry model to be imported.

The **STEP-File Header Data** dialog box will open with settings of geometry model import:



Do not change the settings. Click **OK**.

The program will load the geometry model of the tube:

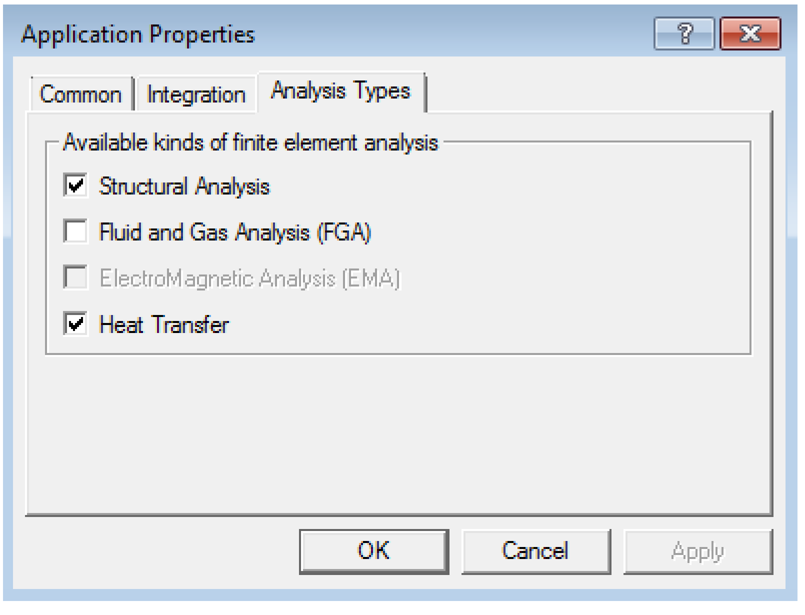


6.5.1.2 Boundary and initial conditions for the thermal calculation

Boundary conditions of heat exchange with surrounding medium

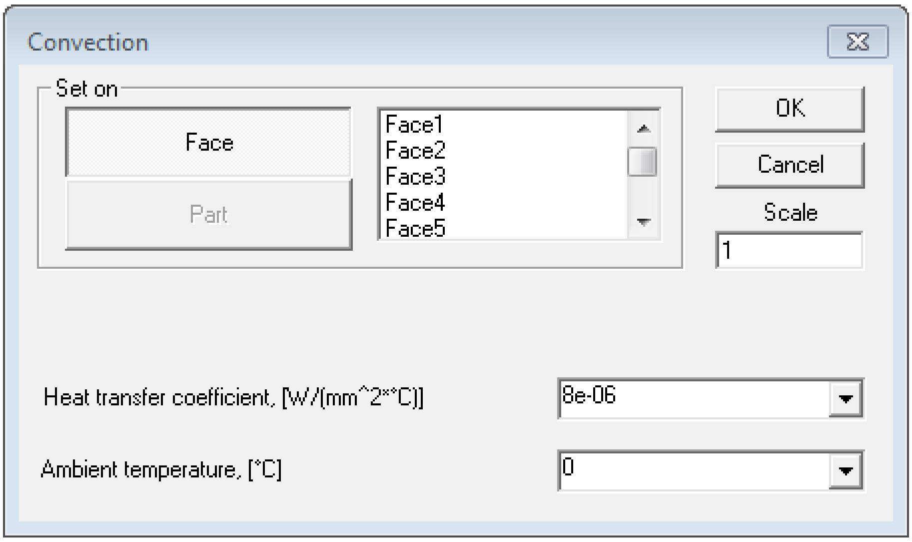
Before specifying the boundary and initial conditions for the thermal calculation, you have to activate the appropriate toolbar.

Apply the command **Tools > Application properties**. The **Application properties** dialog box will open:



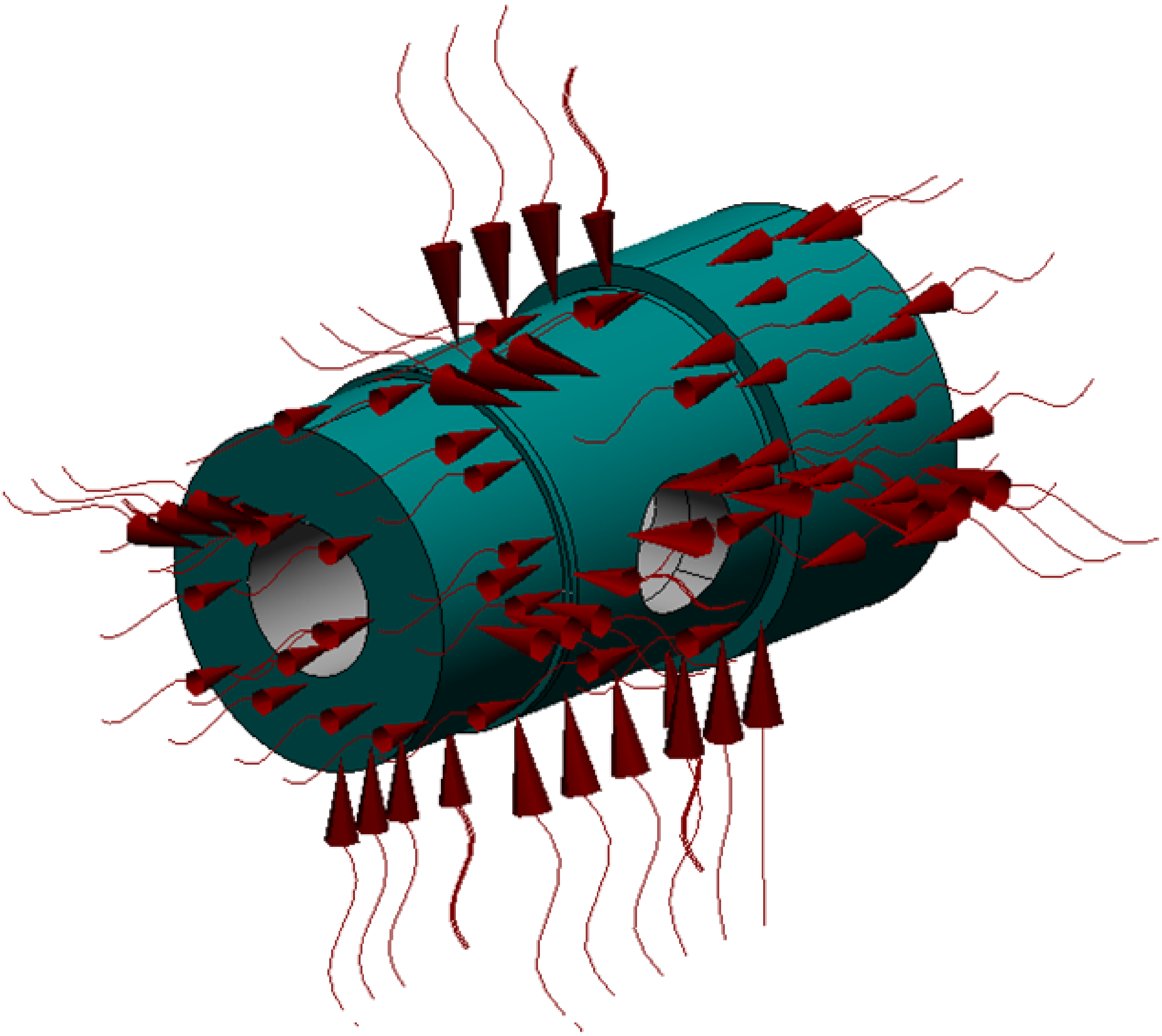
In this dialog box open the tab **Analysis Types** and select the **Heat transfer** checkbox.

You have to specify convection on the outer surface. To do so, select **Transient thermal conductivity > Convection** () in the toolbar. The **Convection** dialog box will open:



In this dialog box specify:

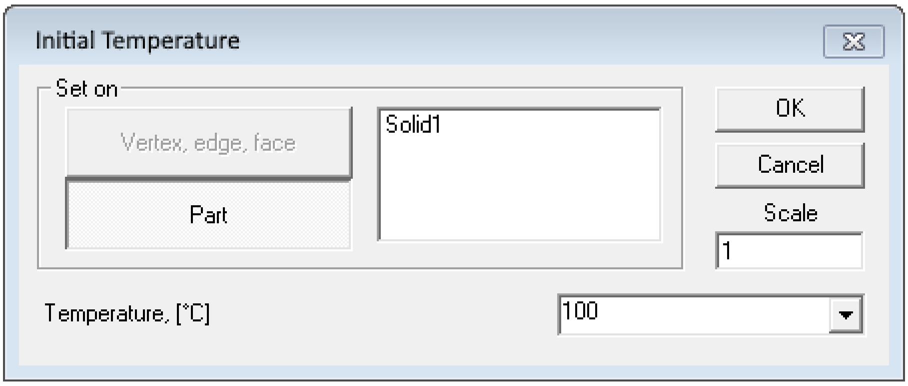
- faces, to which the loads will be applied (select outer faces only)
- **Heat transfer coefficient = 8e-06** [W/(mm^2·°C)]
- **Ambient temperature** = **0** [°C]



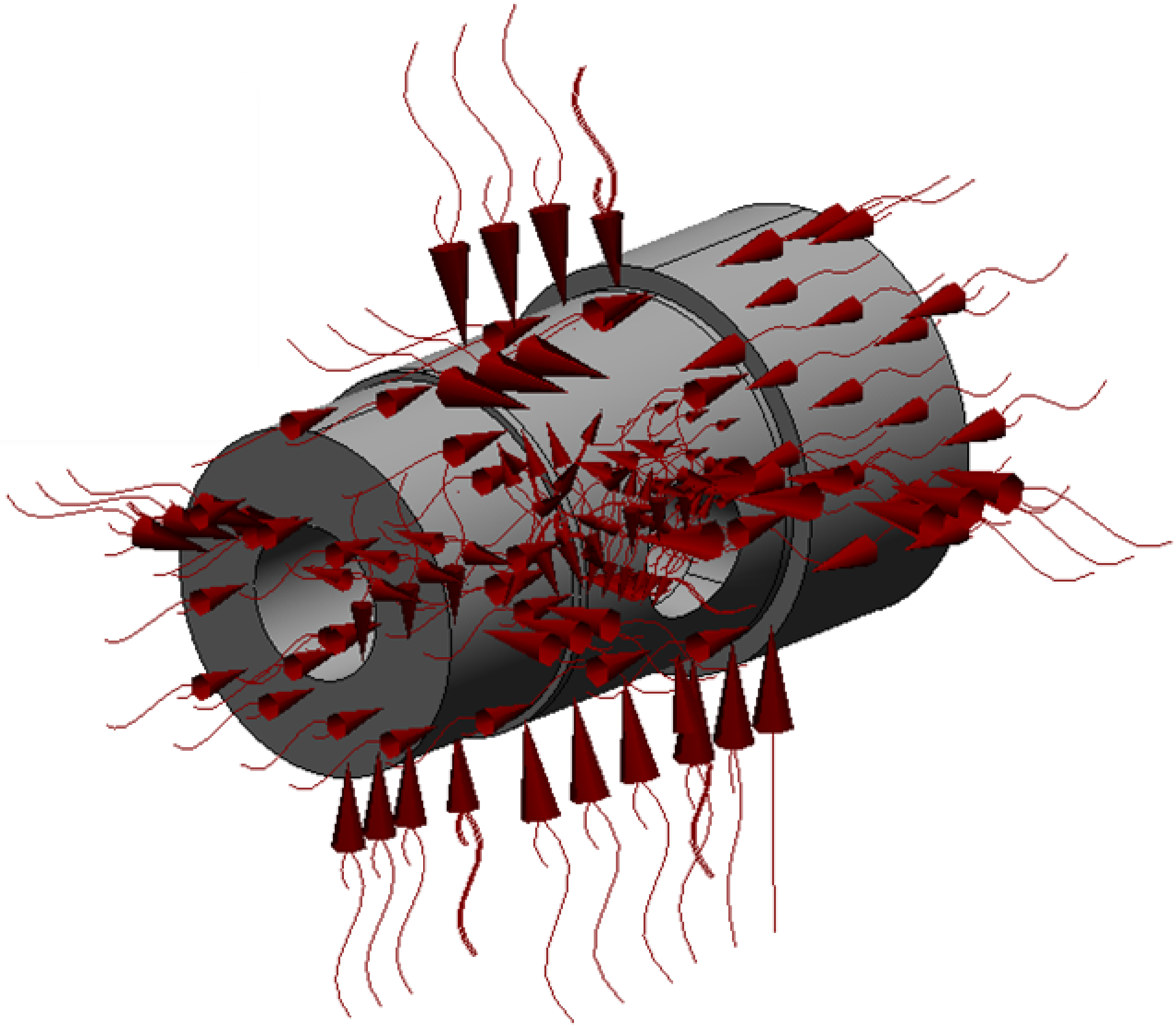
Initial conditions

To carry out an unsteady-state thermal calculation, you have to specify the initial temperature on the whole part.

Select **Transient thermal conductivity > Initial temperature** (). The **Initial Temperature** dialog box will open:

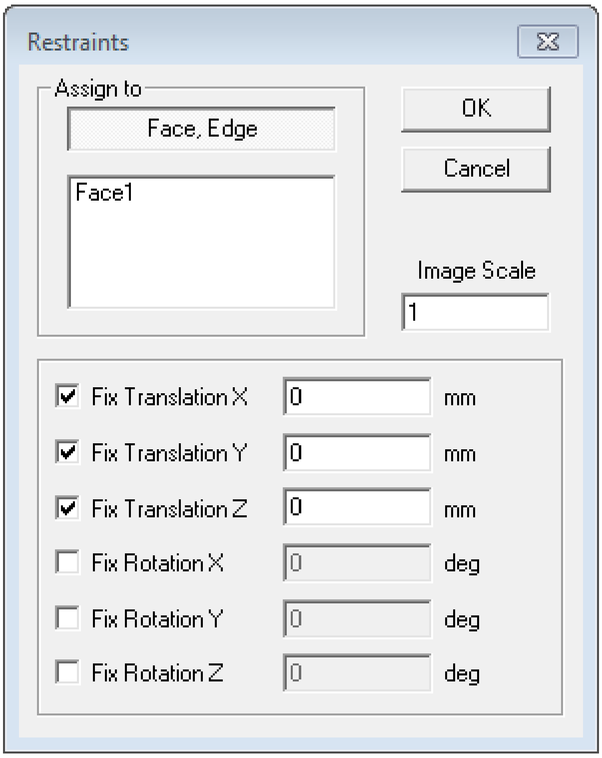


In this dialog box select **Set on > Part** and specify **Temperature = 100** [°C].

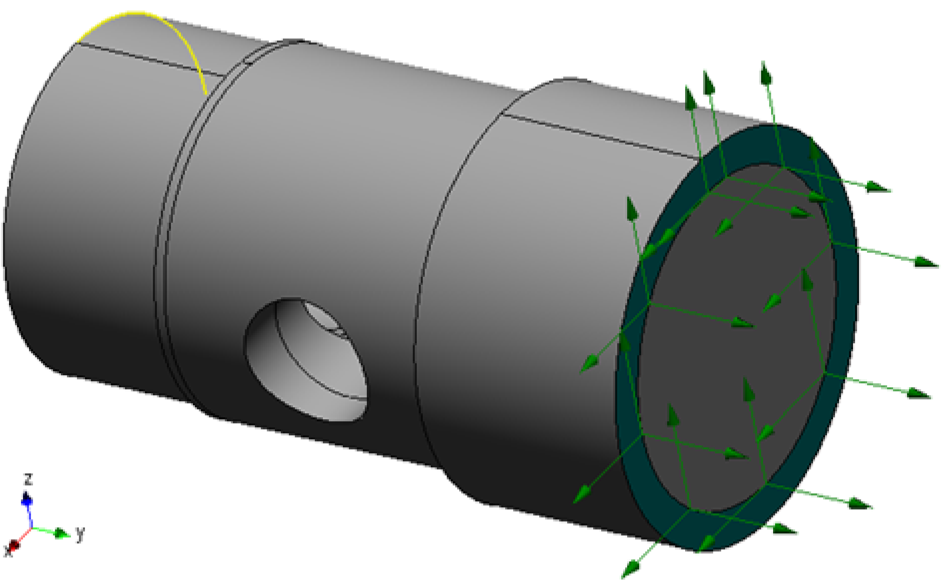


6.5.1.3 Specifying restraints for strength calculation

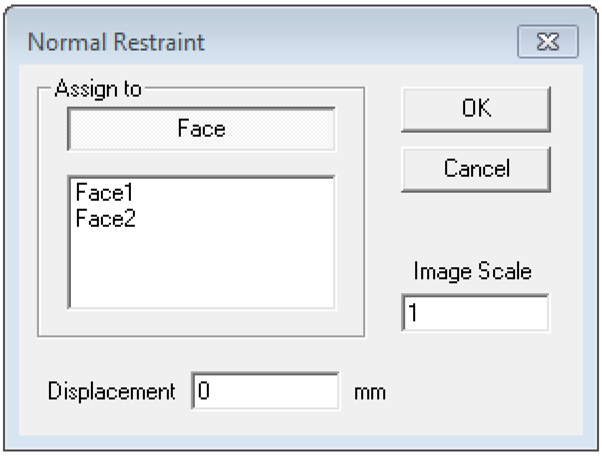
To carry out a strength calculation, you have to secure the part using the command **Fixations > Fixation** (). The **Restraints** dialog box will open:



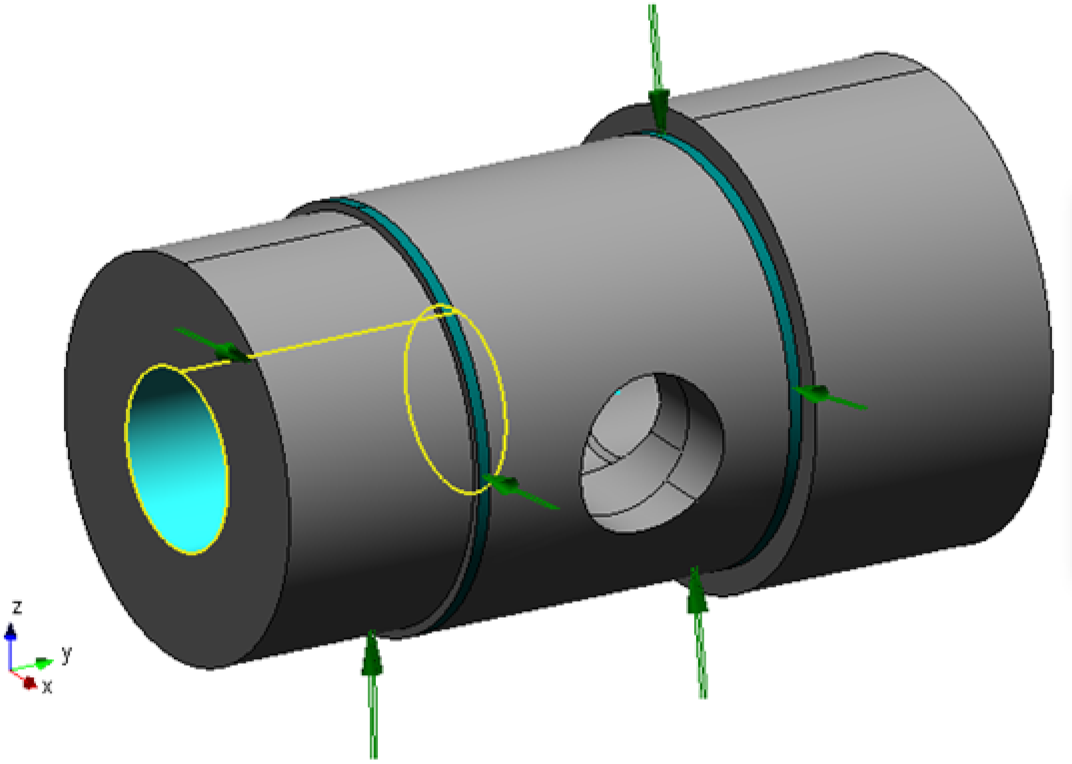
In this dialog box specify the face, which will be restrainted. Select the rim on the upper plane as shown on the illustration below:



Apply the command **Fixations > Normal** (). The **Normal Restraint** dialog box will open:

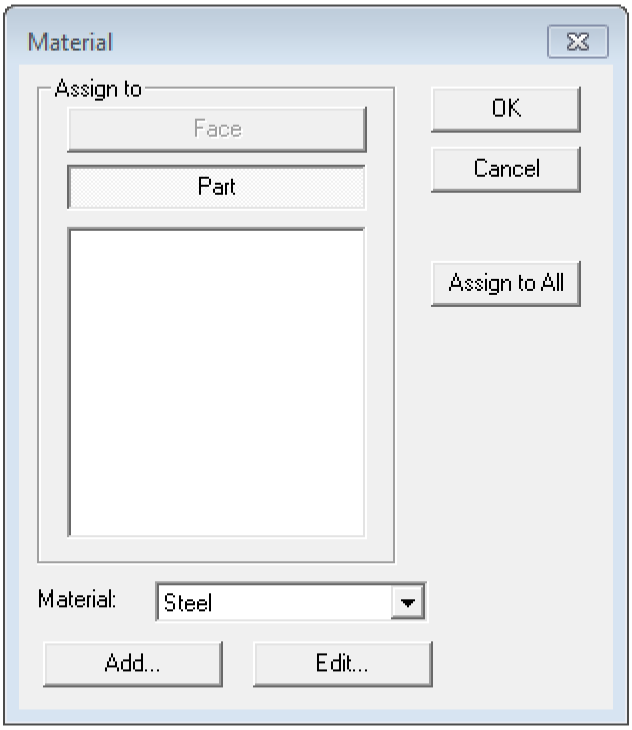


In this dialog box specify the faces that are restrainted along their normals. Do not change the default value of the displacement along a normal, **0** [mm].



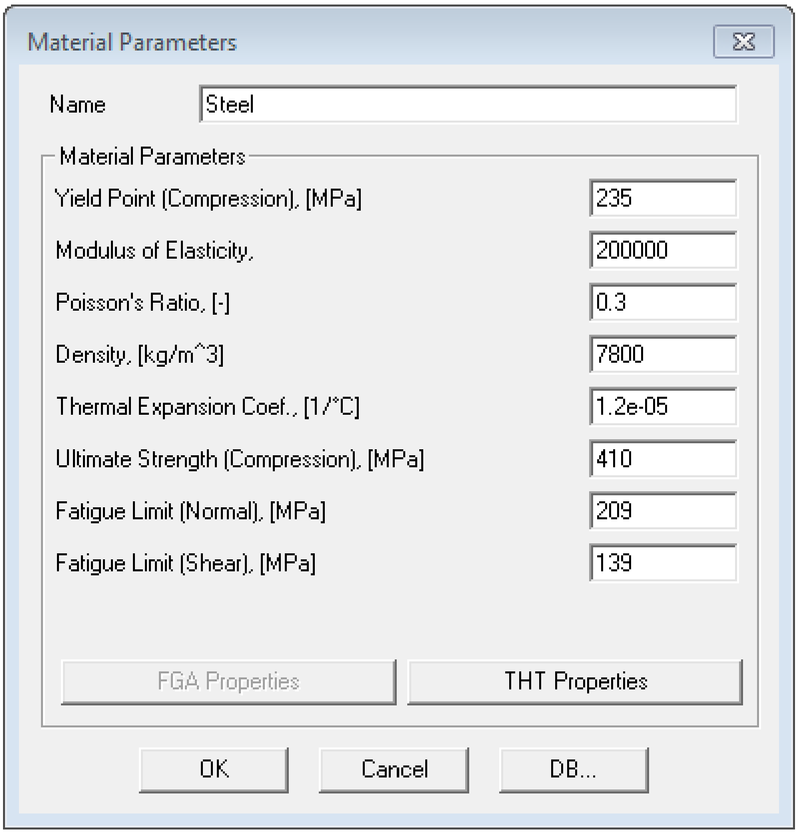
6.5.1.4 Specifying properties of material of the part

To specify properties of the part's material, select **Properties > Material** (). The **Material** dialog box will open:



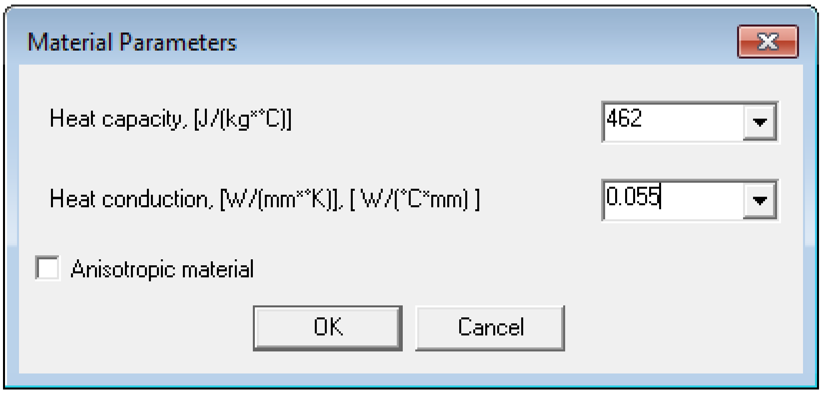
In this dialog box you have to specify properties of the part. Select **Assign to > Part**.

Click the **Edit** screen button. The **Material Parameters** dialog box will open where you have to specify parameters required for the computation.



By default the program uses parameters of **Steel**.

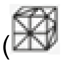
To carry out the thermal calculation, you have to specify thermal properties. Click the **THT Properties** screen button to open the dialog box for entering additional properties:

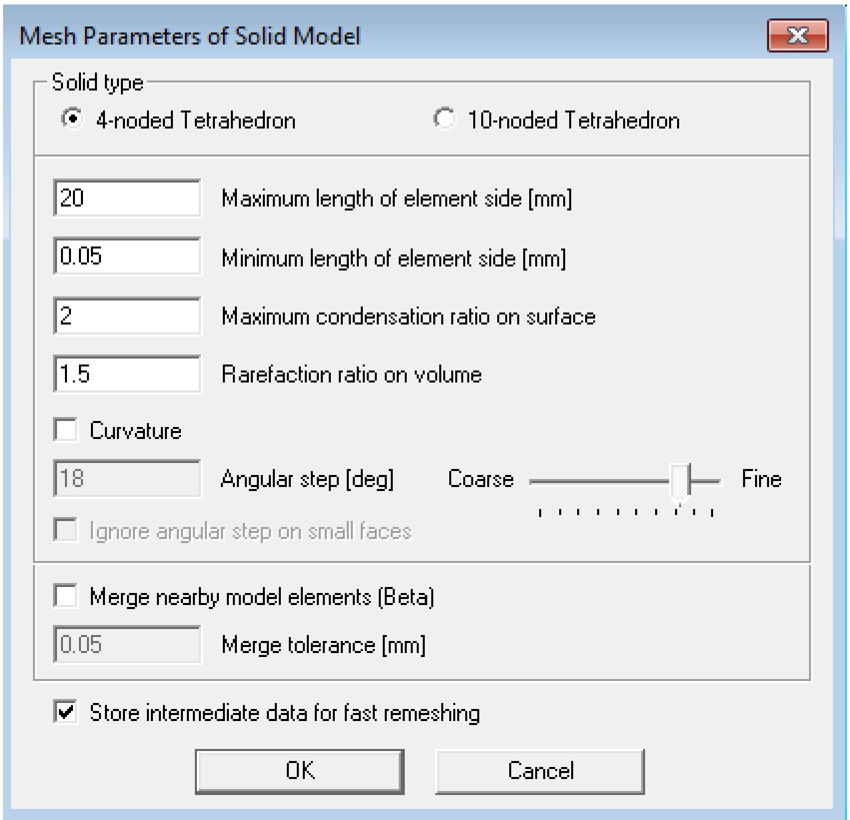


Enter values of **Heat capacity** and **Heat conduction** and click **OK**. Then click **OK** in dialog boxes **Material Parameters** and **Material**.

6.5.1.5 Creating and saving the finite-element mesh

Creating the finite-element mesh

To create the finite-element mesh, select **Finite-element mesh > Generate** () in the toolbar. The **Mesh Parameters of Solid Model** dialog box will open:



In this dialog box specify:

- **Solid type = 4-noded Tetrahedron**
- **Maximum length of element side = 20** [mm]
- **Minimum length of element side = 0.05** [mm]
- **Maximum condensation ratio on surface = 2**
- **Rarefaction ratio on volume = 1.5**

Click **OK**. The **Meshing results** informational box will open. Don't close this box and follow to saving the finite-element mesh (see subsection below).

Saving the finite-element mesh

To save the finite-element mesh, select the command **File > Save FEM**. A dialog box will open where you have to specify a file, in which the *APM Structure3D* project will be saved (enter the name of the project's file).

After saving the file, close *APM Studio*. The further actions will be done in *APM Structure3D*.

6.5.2 Preparing the project in APM Structure3D

APM Structure3D is one of modules of the *APM WinMachine* software package.

APM Structure3D allows you to:

- Prepare a FEA model to computation
- Create an exchange surface for further exchange with *FlowVision*
- Start the prepared in *APM Studio* project to joint computation
- Visualize and process the results

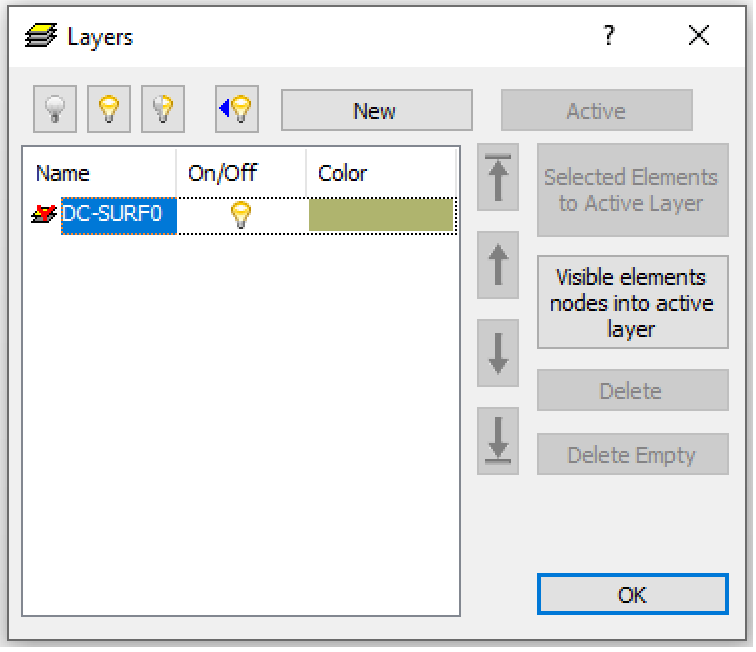
Open *APM Structure3D* and follow steps described below in subsections:

- Naming a layer
- Saving the project and exporting the model

6.5.2.1 Naming a layer

Open the FE mesh in the project using the command **File > Open**. The file is to have extension `.frm` and locate in the same place where it has been saved at saving the finite-element mesh in APM Studio.

Select **Current Parameters > Layers Manager** ( ) from the toolbar. The **Layers** dialog box will open:



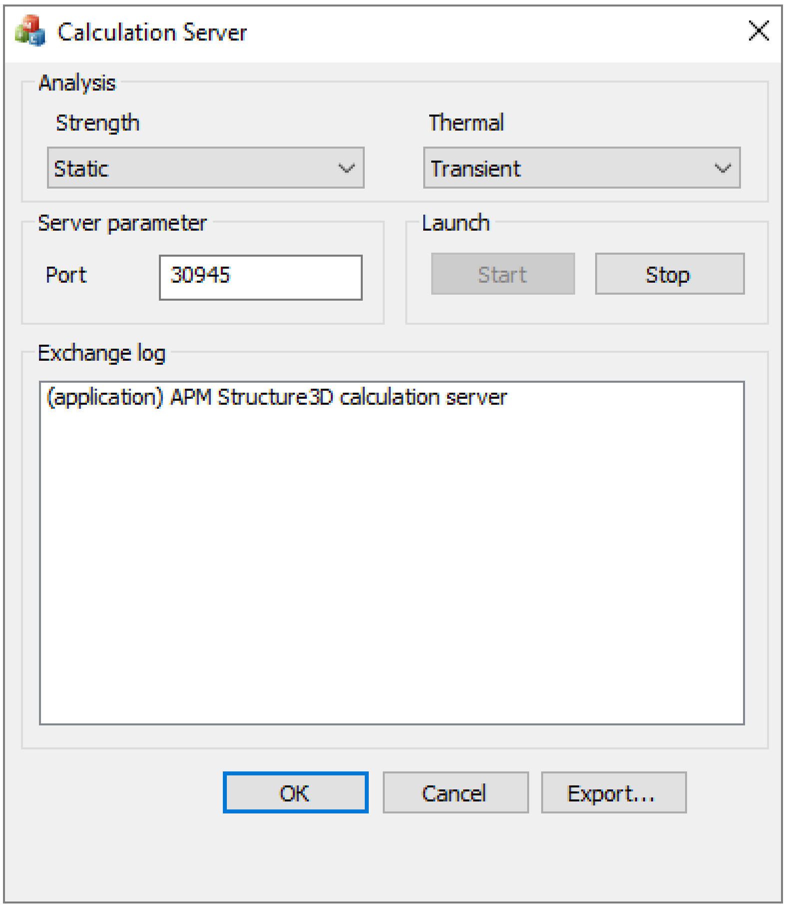
In this dialog box examine number of layers and names of layers. Remove all empty layers. The remaining layer is to be named as **DC-SURF0** (don't change its name).

To rename a layer, you can press the button **F2** on the keyboard.

6.5.2.2 Saving the project and exporting the model

To create a connector in *FlowVision*, you have to export an `.inp` file from *APM Structure3D*.

Apply the command **Tools > Analysis server > Analysis server TCP/IP**. The **Calculation Server** dialog box will open:



In this dialog box specify all required settings and then create a file for data exchange between the programs. To do so, click the **Export** screen button and, in the dialog box, which opens, specify:

- location where the file will be placed
- the file name
- the file type specify as **Flow Vision mesh files (APMFV.inp)**

6.5.3 Preparing the project in FlowVision

Preparing the project in *FlowVision* is described in subsections below:

- Geometry of the region
- Physical model
- Creating the computational domain
- Boundary conditions
- Parameters of co-simulation
- Inspection of the transferred data
- Computational grid
- Parameters of calculation in FlowVision
- Visualization
  - Velocity distribution on a plane
  - Pressure distribution on a plane
  - Temperature distribution on the moving body
  - Integral heat flow rate on the exchange surface

6.5.3.1 Geometry of the region

Open **Pre-Postprocessor** and apply the command **File > Create** from the main menu. In the **New project** dialog box, which opens, select **Create empty project**.

The created **Box** will be the region that contains an inverted **Moving body**. The computational domain will locate inside the **Moving body** only. As shape of the **Moving body** will be changing during the computation, you have to specify the **Box** with some reserve by size.

In the **Geometry** tab in the folder **Initial geom. models** apply the **Create** command from its context menu and create the **Box #0** object and in its properties specify:

Object > Location

Reference point

X	**= 0.15**	[m]
Y	**= -0.1**	[m]
Z	**= 0**	[m]

Object > Size

X	**= 0.75**	[m]
Y	**= 0.9**	[m]
Z	**= 0.3**	[m]

From the context menu of **Box #0** select the command **Create consistent mesh** and in the **Approximation parameters** dialog box, which opens, specify:

Size 1 = 1
Size 2 = 1
Size 3 = 1

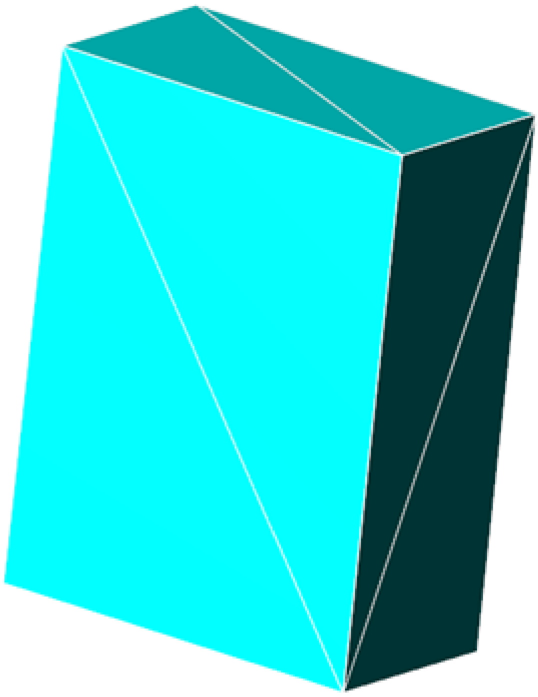
In the project tree the element **Box #0** will obtain a child element **mesh**.

From the context menu of this element **mesh** select the **Use in SubRegion Composer** command. In the subfolder **SubRegion Composer > Objects** the element **Box #0 – mesh** will appear.

From the context menu of this element **SubRegion Composer > Composed subregions** select the **Compose** command. In the subfolder **SubRegion Composer > Composed subregions** the **Box #0 - mesh** child element will appear.

When the composing is done, the **Box #0 - mesh** object can be used as geometry of the **Region** in the **Preprocessor** tab. From the context menu of the element **Composed subregions** select the command **Use as Region main geometry**.

The **Preprocessor** tab will open where the **Subregions** folder will contain a child element corresponding to the **Box #0 - mesh** subregion.



6.5.3.2 Physical model

As during the data transfer *APM* converts all variables into absolute values, you have to specify value of the reference temperature.

It is not allowed the user to specify zero value of the reference temperature in *FlowVision*, so you have to specify small (but not zero) value of it (0.001 K). In properties of the element **General settings** specify the reference values:

- **Temperature = 0.001** [K]
- **Pressure = 101000** [Pa]

In this exercise the **Substance** is water (it is downloaded from the standard **Substance database**).

- Open the context menu of the folder **Substances** and, applying the **Create** command, create **Substance #0**.
- From the context menu of **Substance #0** apply the command **Load from SD > Standard** and download the substance **Water** in its liquid phase. **Substance #0** will change its name to **Water_Liquid** and its child elements (**Molar mass**, **Density**, **Viscosity**, **Thermal conductivity**, ...) will receive their values from the **Substance database**.

In the folder **Phases**:

- Create a continuous **Phase #0**.
- In **Phase #0** add the substance **Water_Liquid** to the subfolder **Phase #0 > Substances**.
- In properties of the folder **Phase #0 > Physical processes** specify:

Heat transfer	**= Heat transfer via h**
Motion	**= Navier-Stokes model**
Turbulence	**= KES**

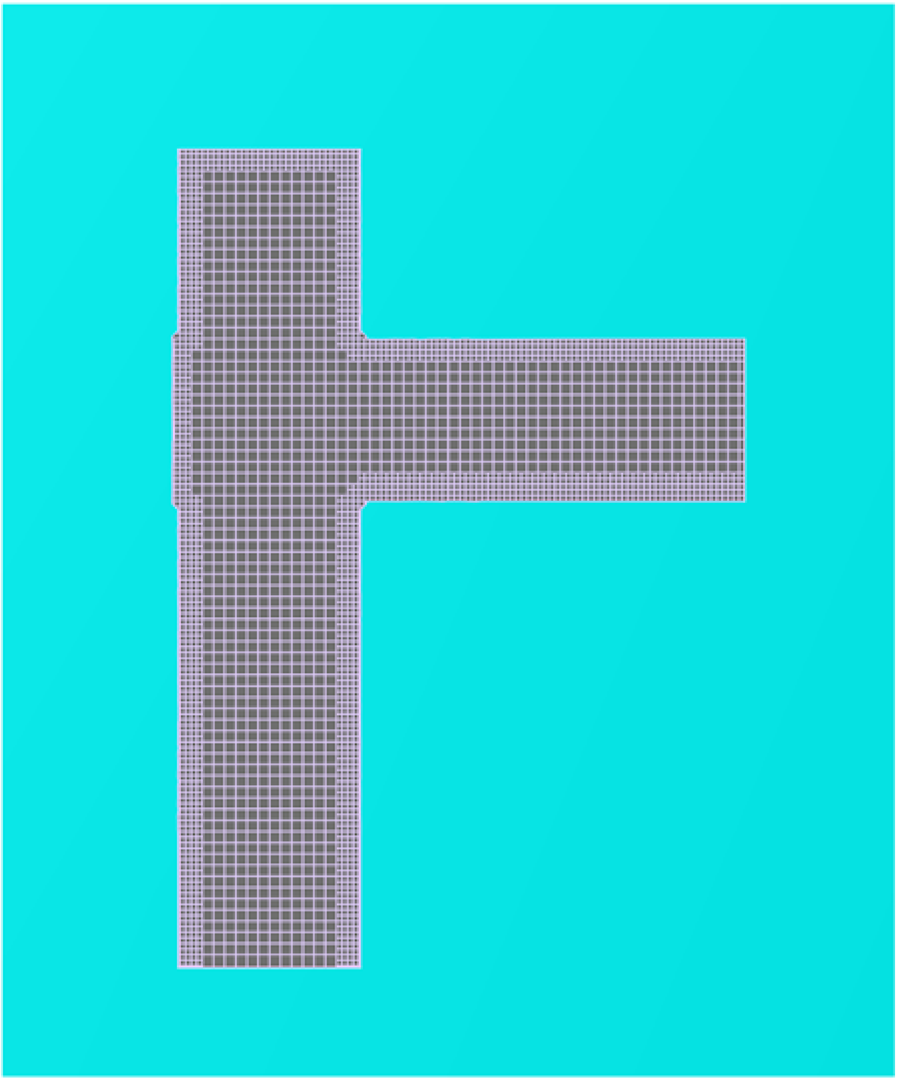
In the folder **Models**:

- Create **Model #0**.
- Add **Phase #0** into subfolder **Model #0 > Phases**.
- Specify initial temperature and pressure:
  - in properties of the element **Models > Model #0 > Init. data > Init. data #0 > Temperature(Phase #0)** specify **Value = 394** [K]
  - in properties of the element **Models > Model #0 > Init. data > Init. data #0 > Pressure(Phase #0)** specify **Value = 69000000** [Pa]

In properties of the element **Subregions > Box #0 - mesh** specify **Model = Model #0**.

6.5.3.3 Creating the computational domain

The computational domain will be created within an inverted **Moving body**.



In the **Preprocessor** tab, from the context menu of the folder **Objects**, select the **Create** command.

The **Create new object** dialog box will open. Specify there **Object type = Imported object** and click **OK**.

Then select the file `ClosedSurf.mesh` that contains geometry of **Imported object**, which will be created.

The program will create **Imported object #0**.

In the folder **Subregions > Box #0 - mesh > Modifiers** create a **Moving body** modifier, specifying in the **Create new modifier** dialog box:

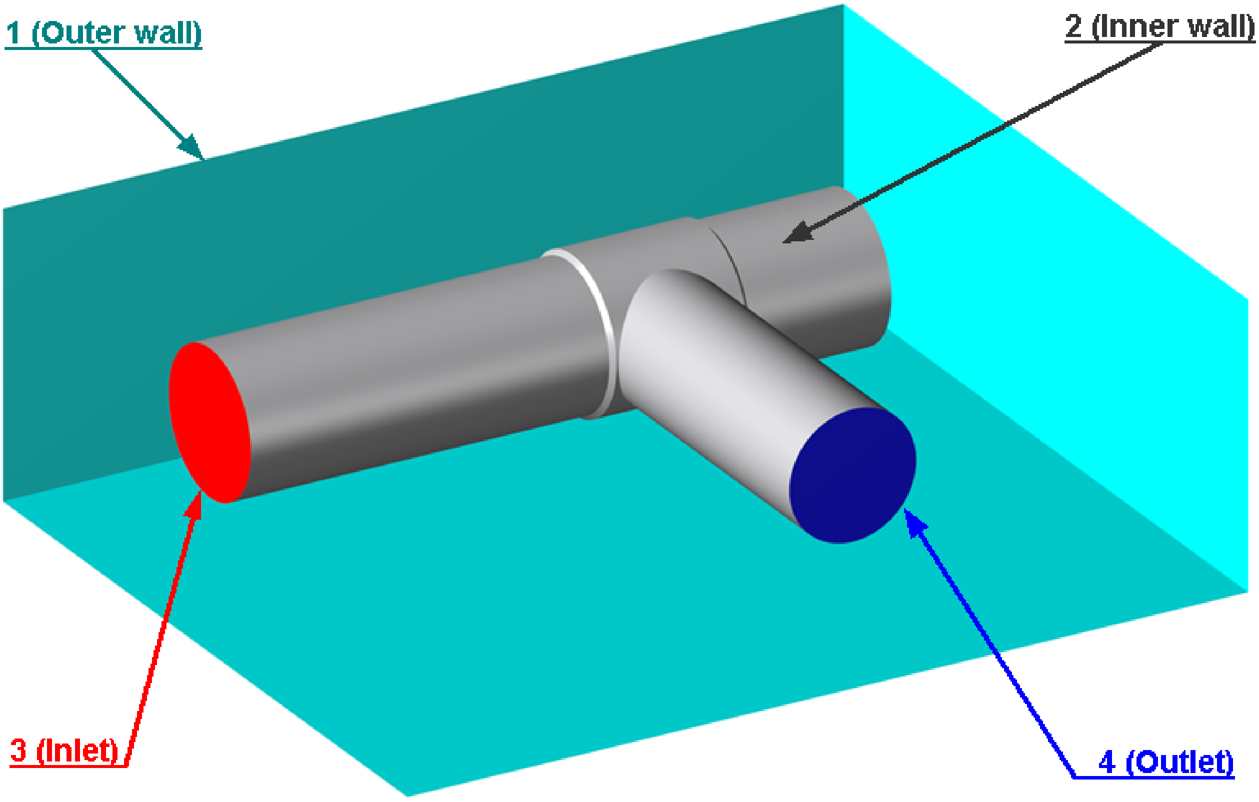
- **Modifier type = Moving body**
- **Objects = Imported object #0**

The program will create **Moving body #0**.

From the context menu of **Moving body #0** select the **Turn inside out** command.

6.5.3.4 Boundary conditions

When inverted geometry is used, the computational domain is created within a **Moving body**.



Specify the boundary conditions:

Boundary 1 (outer wall)

Name	**= Outer wall**
Type	**= Wall**
Variables	
Temperature (Phase #0)	**= Zero gradient**
Velocity (Phase #0)	**= Logarithm law**
TurbEnergy (Phase #0)	**= Value in cell near wall**
TurbDissipation (Phase #0)	**= Value in cell near wall**
Color	= **Aqua**

Boundary 2 (inner wall)

Name	**= Inner wall**
Type	**= Wall**
Variables	
Temperature (Phase #0)	**= External conjugate**
Velocity (Phase #0)	**= Logarithm law**
TurbEnergy (Phase #0)	**= Value in cell near wall**
TurbDissipation (Phase #0)	**= Value in cell near wall**
Color	= **Gray**

Boundary 3 (inlet)

Name	**= Inlet**
Type	**= Inlet/Outlet**

Variables		
Temperature (Phase #0)	**= Temperature**	
Value	**= 394**	[K]
Velocity (Phase #0)	**= Total pressure**	
Value	**= 69000000**	[Pa]
TurbEnergy (Phase #0)	**= Pulsations**	
Value	**= 0**	
TurbDissipation (Phase #0)	**= Turbulent scale**	
Value	**= 0**	[m]
Color	= **Red**	

Boundary 4 (outlet)

Name	**= Outlet**	
Type	**= Inlet/Outlet**	
Variables		
Temperature (Phase #0)	**= Temperature**	
Value	**= 394**	[K]
Velocity (Phase #0)	**= Total pressure**	
Value	**= 68000000**	[Pa]
TurbEnergy (Phase #0)	**= Pulsations**	
Value	**= 0**	
TurbDissipation (Phase #0)	**= Turbulent scale**	
Value	**= 0**	[m]
Color	= **Blue**	

6.5.3.5 Parameters of co-simulation

Joint computations of *APM* and *FlowVision* use the **Abaqus Direct Coupling** connector. This connector allows the software to transfer data directly from one program to another and it is convenient for use.

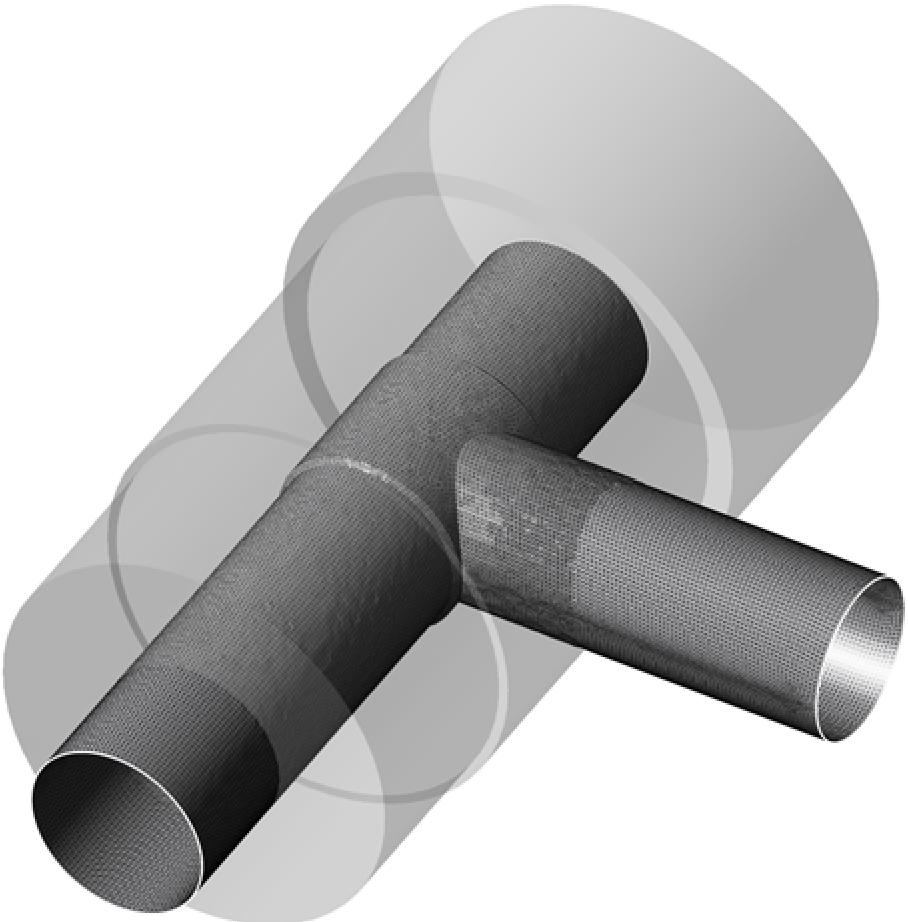
- From context menu of the folder **External Connections** select the command **Create**.
- In the **Create new object** dialog box, which opens, select **Object type = Abaqus Direct Coupling**.
- A dialog box for selection an `inp` file will open. Select there the saved before file **`APMFV.inp`**.
- In the **Editing .inp file** dialog box select the checkboxes **CF**, **CFL**, **COORD**, and **NT**.
- A child element **Abaqus Direct Coupling** will appear in the folder **External Connections**. The element **Abaqus Direct Coupling**, in its turn, also has a child element, **ASSEMBLY_DC-SURF0**.
- In properties of the folder **External Connections** specify:
  Exchange step > Method = FlowVision
  Exchange step > Default time step = 0.005
  Exchange step > Coef. for time step = 1
  Old offset = Yes
- In properties of the element **Abaqus Direct Coupling** specify:
  Activation = Yes
  ABAQUS > Run ABAQUS = No
  ABAQUS > IP Source = User
  ABAQUS > Address = *network name or IP-address of the computer, on which APM Structure3D will run*
  ABAQUS > Port = 30945
  ABAQUS > Timeout [s] = 100
  Loads relaxation > Scale factor = 1
  Loads relaxation > Start in steps = 0
  Loads relaxation > End in steps = 100

Loads relaxation > Initial coefficient = 0
Loads relaxation > Final coefficient = 1
Heat relaxation > Scale factor = 1
Heat relaxation > Start in steps = 0
Heat relaxation > End in steps = 0
Heat relaxation > Initial coefficient = 1
Heat relaxation > Final coefficient = 1

- In properties of the element **ASSEMBLY_DC-SURF0** specify:
  Name > DC-SURF0
  Moving body = Moving body #0
  Mapping > Enabled = Yes
  Mapping > Surface = Initial
  Mapping > Rebuild mapper = Yes

Do not change other settings.

The mapping is used to map the mesh from the CFD surface to the FEA mesh. See details about this in the *FlowVision*'s "*User's guide*" document (section *Mapping*).



6.5.3.6 Inspection of the transferred data

You have to check that all data are transferred.

The imported and exported data are shown in properties of the element **ASSEMBLY_DC-SURF0** in fields **Send data** and **Receive data**.

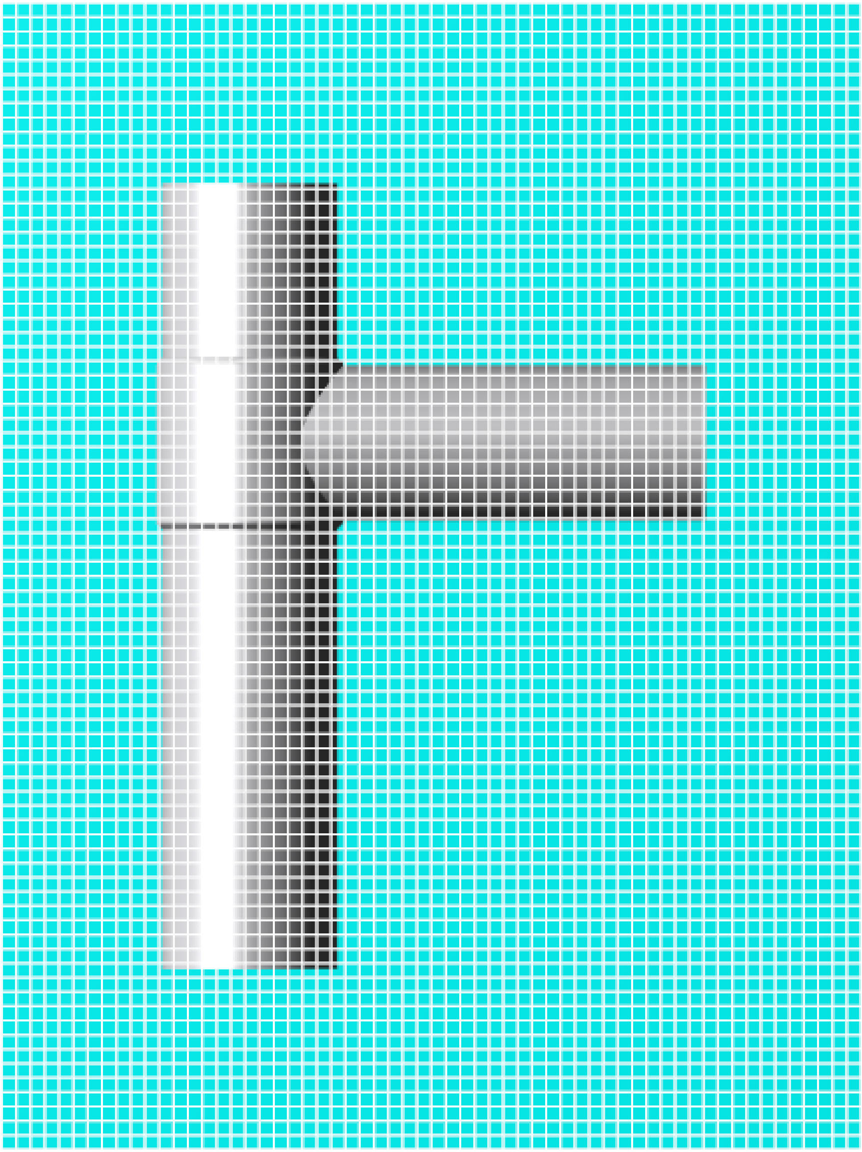
FlowVision exports concentrated force (**CF**) and heat flux (**CFL**), and imports coordinates (**COORD**) and temperature from nodes (**NT**).

6.5.3.7 Computational grid

Initial grid

Set uniform initial computational grid. In properties of the element **Computational grid > Initial grid** specify:

nX=80
nY=96
nZ=32



Adaptation of the initial grid

You have to specify refining of the computational grid near deformable walls. Follow these steps:

- From the context menu of the folder **Computational grid > Adaptation** select the **Create** command and create the element **Adaptation #0**.
- In properties of the element **Adaptation #0** specify:
  Max level N = 1
  Split/Merge = Split
  Layers > Layers for Level N = 4

The subregion **Box #0 - mesh** will be automatically set as the **Subregion**, in which the adaptation will be active.

As an **Object** for the adaptation you have to select the boundary condition **Inner wall** (from the context menu of the folder **Computational grid > Adaptation > Adaptation #0 > Objects** select the command

Add/Remove Boundary Conditions and in the dialog box, which opens, select the boundary condition **Inner wall**).

6.5.3.8 Parameters of calculation in FlowVision

Specify parameters of calculation of *FlowVision* in the **Solver** tab.

To provide stable solution and convergence, CFL numbers and limiter on the maximal time step are specified by tables.

In the **Solver** tab, in properties of the element **Time step**, specify:

Method	**= Via CFL number**
Convective CFL	Specify it by a table that contains dependency on the **Current time step** (see below)
Surface CFL	**= 1**
Diffusive CFL	Specify it by a table that contains dependency on the **Current time step** (see below)
Max step	Specify it by a table that contains dependency on the **Current time step** (see below)

Dependency of **Convective CFL** on the **Current time step**	
x	**f(x)**
0	**1**
20	**1**
50	**100**
70	**100**
100	**500**
120	**500**
150	**2000**
170	**2000**
200	**5000**
220	**5000**
250	**7500**
270	**7500**
300	**10000**
320	**10000**
400	**12000**

If you wish, you can import this table from the text file `ConvCFL.txt` that is included in files of this exercise.

Dependency of **Diffusive CFL** on the **Current time step**	
x	**f(x)**
0	**1**
20	**1**
50	**10**
70	**10**
100	**50**
120	**50**
150	**200**
170	**200**
200	**500**
220	**500**
250	**750**
270	**750**
300	**1000**
320	**1000**
400	**1200**

If you wish, you can import this table from the text file `DiffCFL.txt` that is included in files of this exercise.

Dependency of **Max step** on the **Current time step**	
x	f(x)
0	0.001
50	0.001
100	1
200	1
250	5
350	5
400	10

If you wish, you can import this table from the text file `MaxStep.txt` that is included in files of this exercise.

Set the limitation **50000** [W/m^2] on the maximal energy flow transferred to the FEA program.

In the **Solver** tab, in properties of the element **Advanced settings** specify:

Computation of loads

Energy flux max. = **50000** [W/m^2]

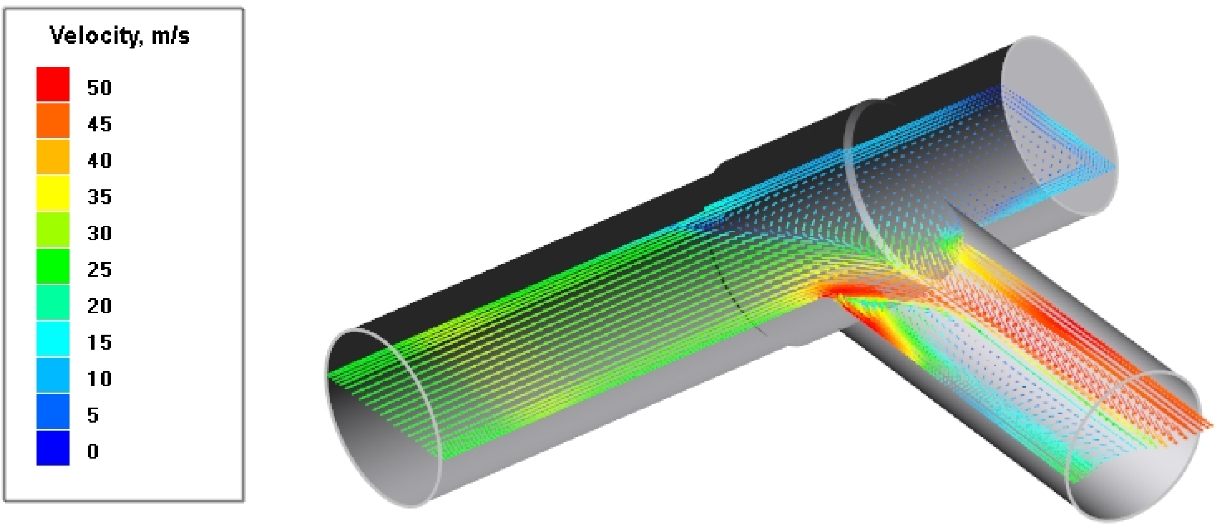
6.5.3.9 Visualization

Specifying the visualization in *FlowVision* is described below in sections:

- Velocity distribution on a plane
- Pressure distribution on a plane
- Temperature distribution on the moving body
- Integral heat flux over the exchange surface

You can view the prepared visualization layers and plot in the **Monitor** window after starting the computation.

6.5.3.9.1 Velocity distribution on a plane



- In properties of **Plane #0** specify:

Object

Reference point

X = **0**

Y = **0**

Z = **0**

Normal

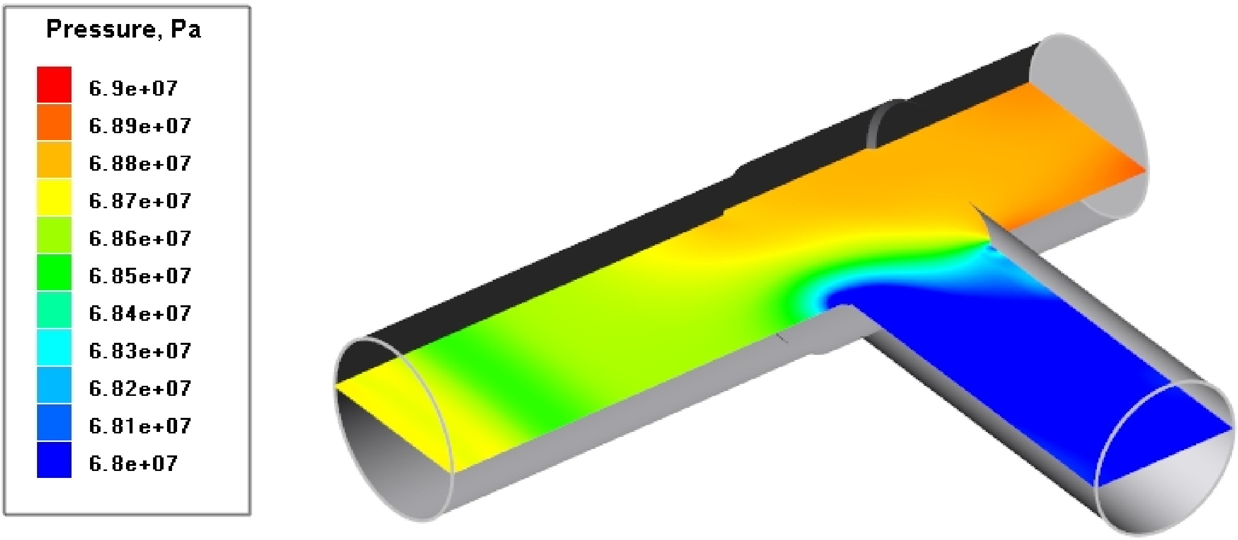
X **= 0**
Y **= 0**
Z **= 1**

- Create a layer **Vectors #0** on **Plane #0**.
- In properties of the layer **Vectors #0** specify*):

On regular grid **= No**
Coloring
Variable
Variable **= Velocity**
Value range
Mode **= Manual**
Max **= 50**
Min **= 0**

*) The program automatically specifies the variable that is used to build vectors (as if you set **Variable > Variable = Velocity**).

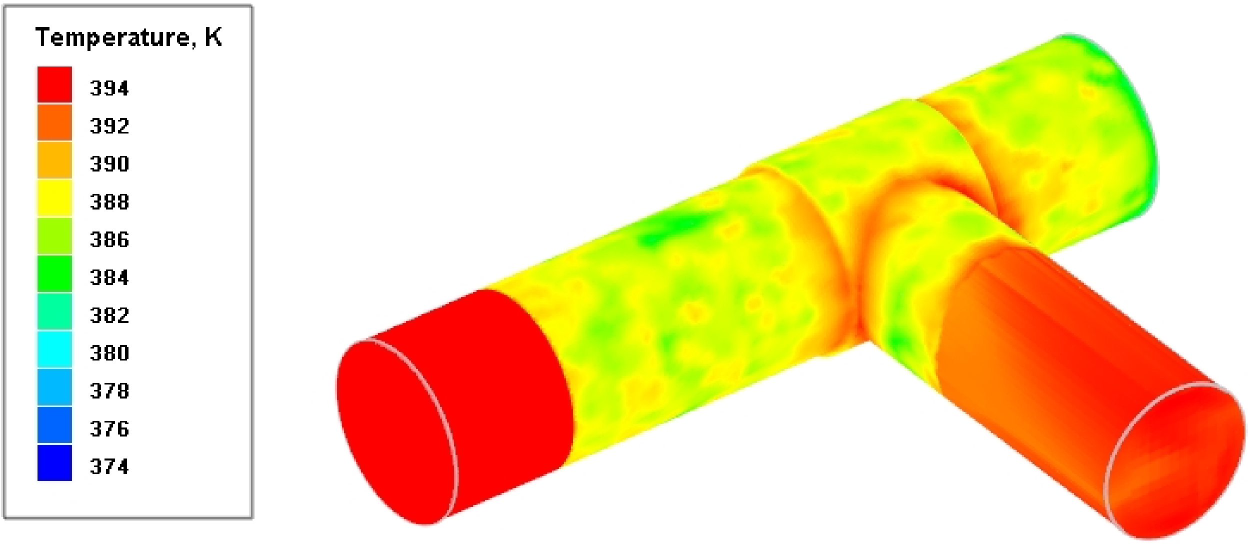
6.5.3.9.2 Pressure distribution on a plane



- Create a **Color contours** layer on **Plane #0**.
- In properties of the created layer **Color contours #0** specify:

Variable
Variable **= Pressure**
Value range
Mode **= Manual**
Max **= 69000000**
Min **= 68000000**

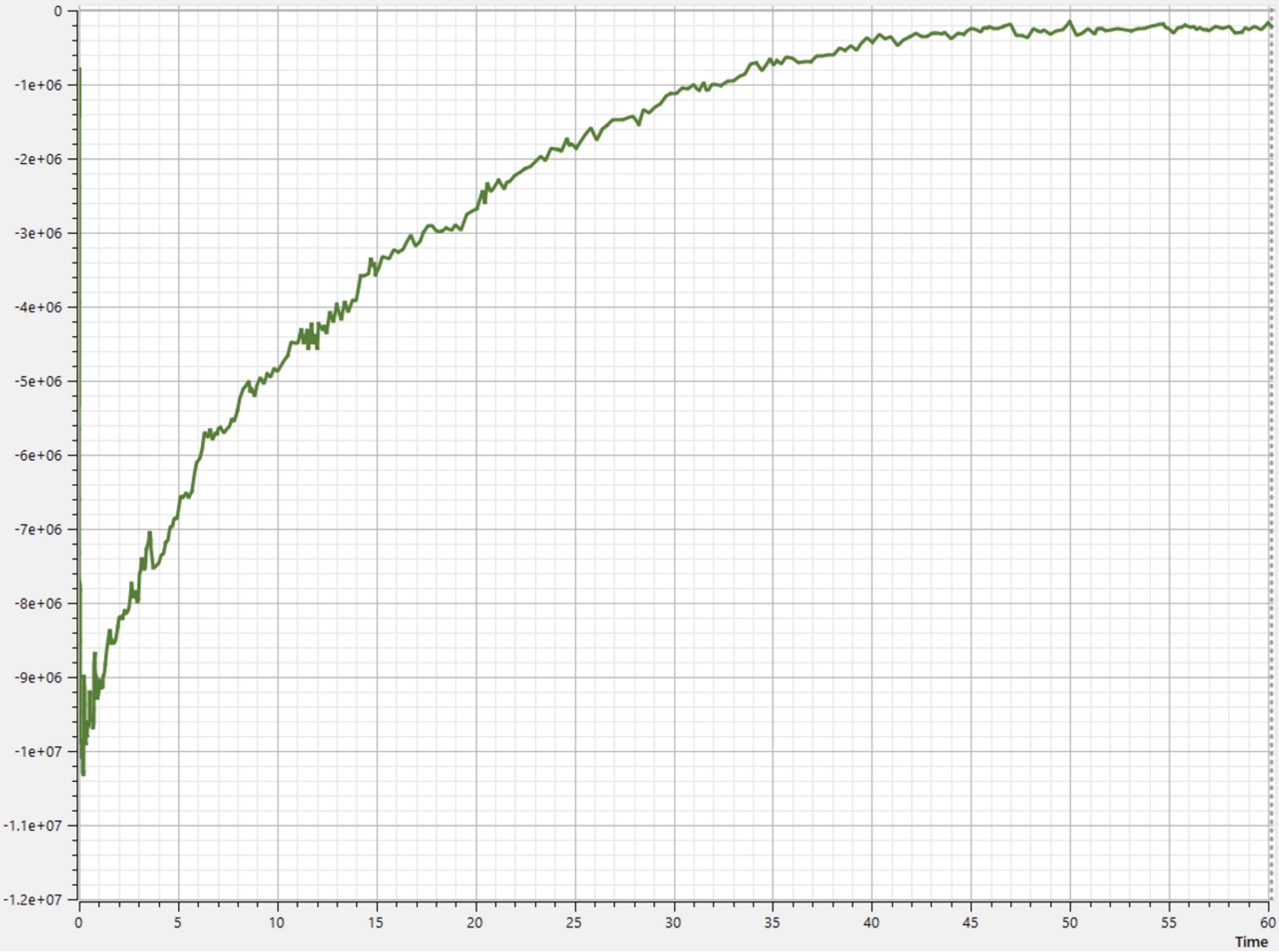
6.5.3.9.3 Temperature distribution on the moving body



- Create a **Color contours** layer on the **Imported object #0**.
- In properties of the created layer **Color contours #1** specify:

Name		**= Temperature**
Variable		
	Variable	**= Temperature**
Value range		
	Mode	**= Manual**
	Max	**= 394**
	Min	**= 374**

6.5.3.9.4 Integral heat flux over the exchange surface



Convergence criterion for this problem is integral heat flux that is calculated on the exchange surface. When the heat flux becomes constant, we can consider that the steady-state is obtained. This value will not be zero because heat transfer from the part's wall to the environment is set, so *FlowVision* will show constant negative value of the heat flux.

We can display this criterion in the **Monitor** window as a plot of the time dependance.

Follow these steps:

- On the boundary condition **Inner wall** create a **Supergroup** in **Preprocessor**.
- In the folder **Characteristics** in **Preprocessor** create **Characteristics** on the **Supergroup** on the **Inner wall**.
- In properties of the just created element **Characteristic #0 (Supergroup on "Inner wall")** specify **Variable > Variable = HeatFlux**.
- Open the **Solver** tab.
- From the context menu of the folder **Stopping conditions > User values** select the command **Create** and create **Stop criterion #0**.
- In properties of the just created **Stop criterion #0** specify:
  - **Name = <Heat Flux>**
  - **Object = Characteristics #0 (Supergroup on "Inner wall")**
  - **Variable = <f surf.>**

6.5.4 Starting the joint computation

It doesn't matter, in which order the two programs will be started.

In *APM Structure3D* to control the joint computation use the **Exchange Log** block in the **Calculation Server** dialog box of the *APM Structure3D* program. Here you can see all information about the current status of the computation.

In *FlowVision* use windows **Log** and **Monitor** in **Pre-Postprocessor**.

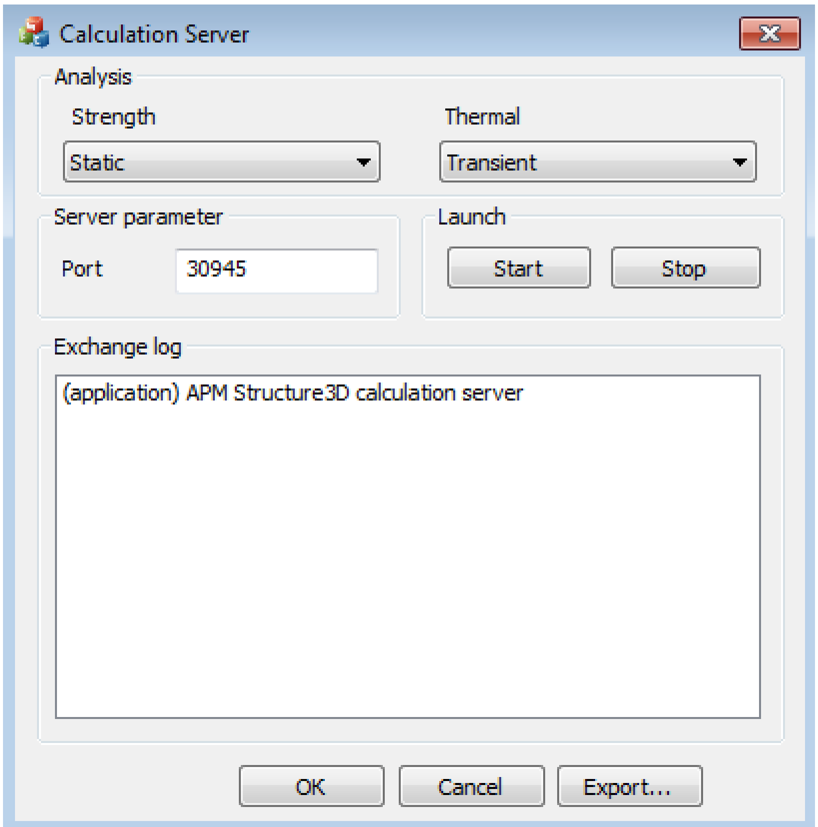
Starting the computation from FlowVision

In *FlowVision* starting the computation is done from **Pre-Postprocessor**, see description of the standard start in the section "Detailed description of a simplest model > Laminar flow in a tube > Starting the computation".

You can stop the computation by clicking the (**Stop computation**) button and resume it after changing the project if required.

Starting the computation from APM Structure3D

In *APM Structure3D* select **Tools > Analysis server > Analysis server TCP/IP**. The **Calculation Server** dialog box will open:



Specify the following settings:

- In the block **Analysis** specify, which computations will be done in *APM Structure3D* in the joint computation. Specify here:
  - **Strength = Static**
  - **Thermal = Transient**
- In the block **Server parameter** you have to examine the value of the parameter **Port**. This value is to be the same as the port number that is set in FlowVision in properties of the element "External Connections > Abaqus Direct Coupling". By default **Port=30945**.

> Parameters of linear and geometrically non-linear computations, steady-state and transient heat transfer can be set in the **Calculation > Calculation Options Panel** menu.
>
> You can find information about the calculation parameters in the user's manual of *APM Structure3D*.

Start or stop of the data exchange between the programs is done by clicking buttons **Start** or **Stop** in the **Calculation Server** dialog box, in the **Launch** block:

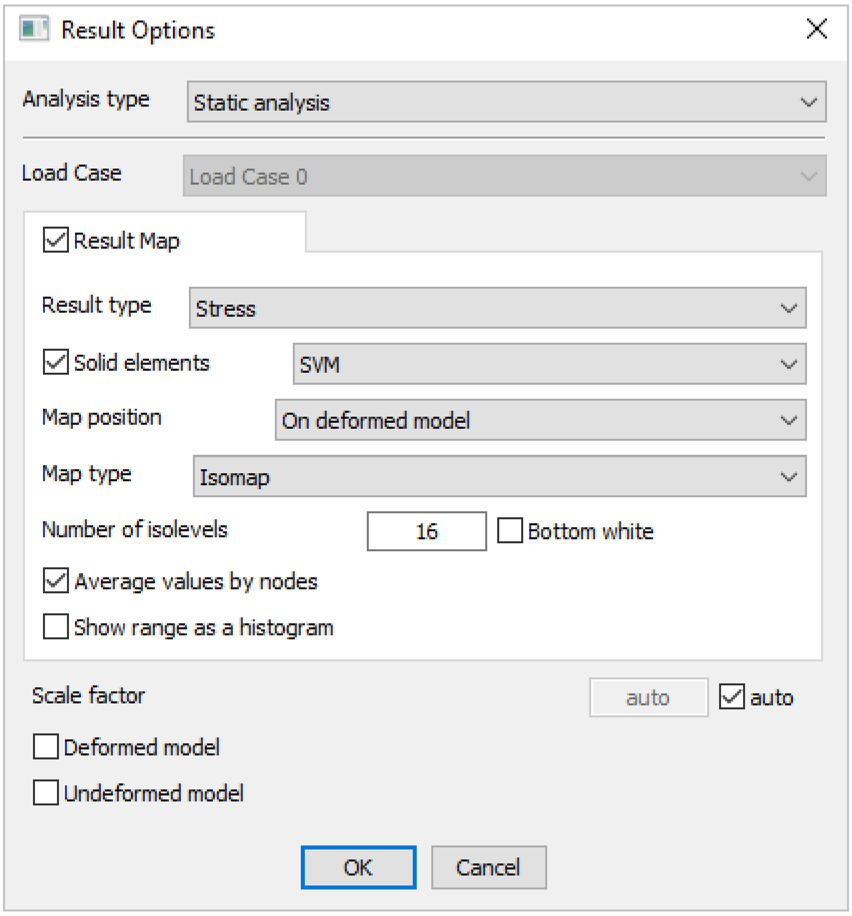
Viewing results in FlowVision

In **Pre-Postprocessor** watch the prepared before visualization layers and the plot:

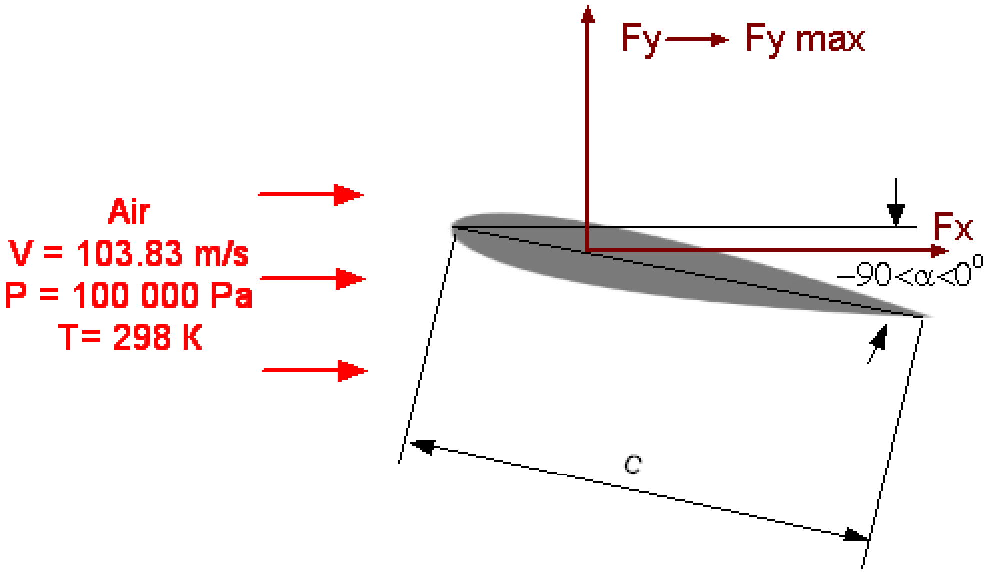
* Velocity distribution on a plane
* Pressure distribution on a plane
* Temperature distribution on the moving body
* Integral heat flux over the exchange surface

Viewing results in APM Structure3D

When the computation is finished, select the command **Result Map** from the menu **Results**. In the **Results Options** dialog box, which opens, select the required maps for the type of computation, in which you are interested in:



6.6 Optimization of an airfoil's orientation



In this example we we consider optimization of orientation of the *NACA0012* profile.

The goal of the optimization (optimization criterion) is maximal lift force acting on the profile.

The adjustable parameter (the optimization parameter) is the angle between the profile and the flow direction (angle of attack).

Parameters of the problem setting

Dimensions:			
Length of the airfoil chord	c	= 0.256	[m]
Dimensions of the computational domain		5.3 × 5 × 0.00254	[m × m × m]
Substance:		Air	
Inlet parameters:			
Static pressure:	P	= 100000	[Pa]
Static temperature:	T	= 298	[K]
Velocity on inlet	V_{inl}	= 103.83	[m s^{-1}]
Mach number	M	= 0.3	
Reynolds number	Re	= 1.68x 10^6	
Geometry:	`NACA0012_opt.wrl`		
Project:	`NACA0012_opt`		

6.6.1 Preparing the project in FlowVision

6.6.1.1 Physical model

In the **Properties** window of the element **General settings** specify:

Reference values

Temperature	**= 298**	[K]
Pressure	**= 100000**	[Pa]

In the folder **Substances**:

- Create **Substance #0**.
- Load **Substance #0** from the **Standard** substance database:

- From the context menu of **Substance #0** select **Load from SD > Standard**.
- In the new window **Load from database**, select:

Substances	**= Air**
Phases	**= Gas (equilibrium)**

In the folder **Phases**:

- Create a continuous **Phase #0**.
- Add the **Air_Gas (equilibrium)** substance into the folder **Phase #0 > Substances**.
- Specify in properties of the folder **Physical processes**:

Motion	**= Navier-Stokes model**
Heat transfer	**= Heat transfer via H**
Turbulence	= **KES** *)

*) This means the *Standard k-ε turbulence model*.

In the folder **Models**:

- Create **Model #0**.
- Add **Phase #0** into subfolder **Model #0 > Phases**.
- In the folder **Init. data #0**, specify:

Velocity

X	**= 103.83**	[m s^{-1}]
Y	**= 0**	[m s^{-1}]
Z	**= 0**	[m s^{-1}]

6.6.1.2 Moving bodies

In the **Properties** window of **SubRegion #0**, specify:

Model = Model #0

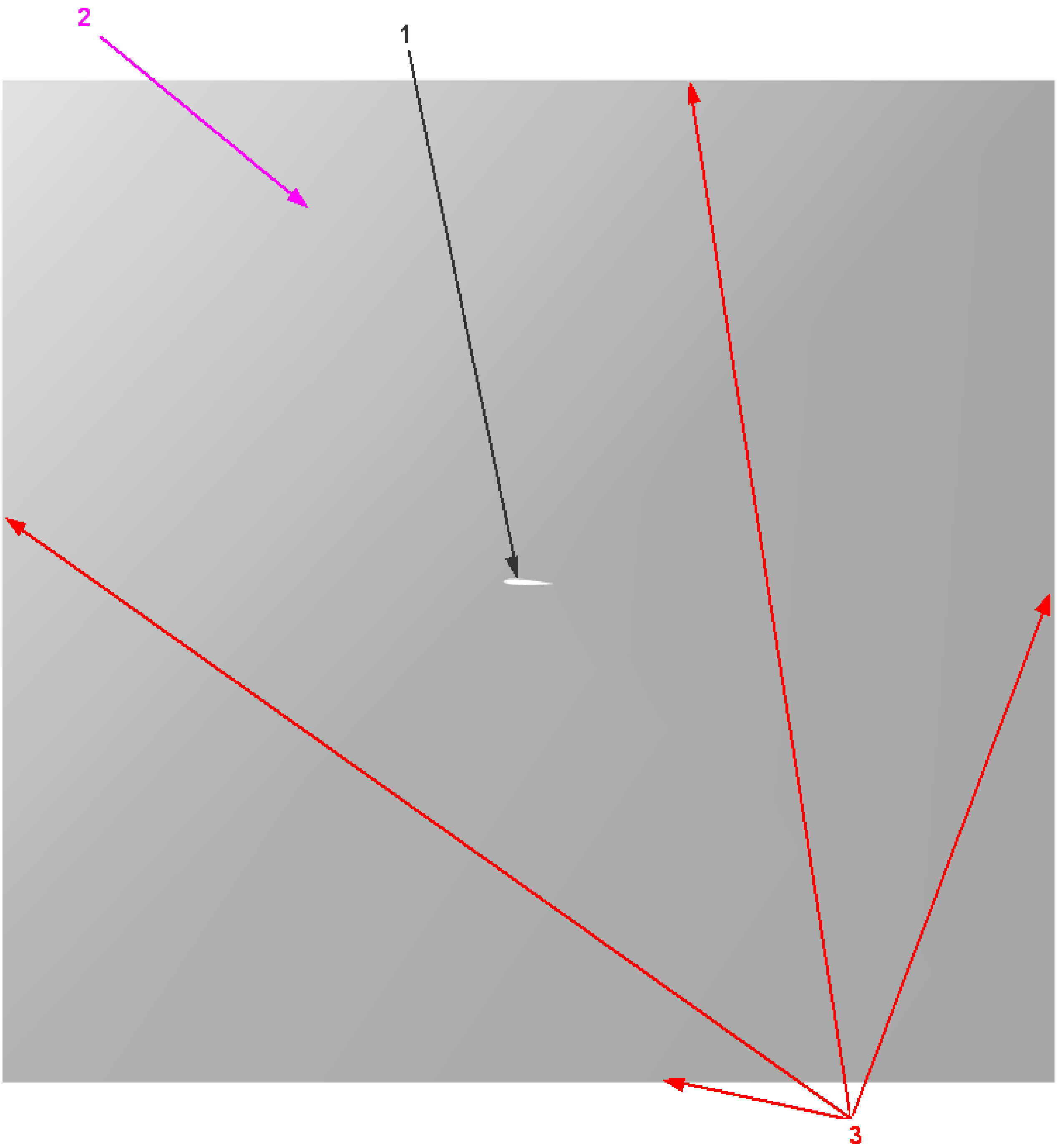
Create a **Moving body**:

- Download the geometry of the moving body from the file `NACA0012_opt_Airfoil.wrl`.
- In the folder **SubRegion #0 > Modifiers** create a **Moving body** modifier on **Imported object #0**.
- In the **Properties** window of the just created **Moving body**, specify:

Initial position

Rotation definition	**= Euler angles (Aviation)**

6.6.1.3 Boundary conditions



Specify the following boundary conditions:

Boundary 1

Type	**= Wall**
Variables	
Temperature(Phase #0)	**= Zero gradient**
Velocity(Phase #0)	**= Logarithm law**
TurbEnergy(Phase #0)	**= Value in cell near wall**
TurbDissipation(Phase #0)	**= Value in cell near wall**

Boundary 2

Type = **Symmetry**

Variables

Temperature(Phase #0) = **Symmetry**

Velocity(Phase #0) = **Slip**

TurbEnergy(Phase #0) = **Symmetry**

TurbDissipation(Phase #0) = **Symmetry**

Boundary 3

Type = **Non-reflecting**

Variables

Temperature(Phase #0) = **Non-reflect.**

Value = **0** [K]

Velocity(Phase #0) = **Non-reflect.**

Velocity at inf.

X = **103.83** [m s^{-1}]

Y

Z

Pressure at inf. = **0** [Pa]

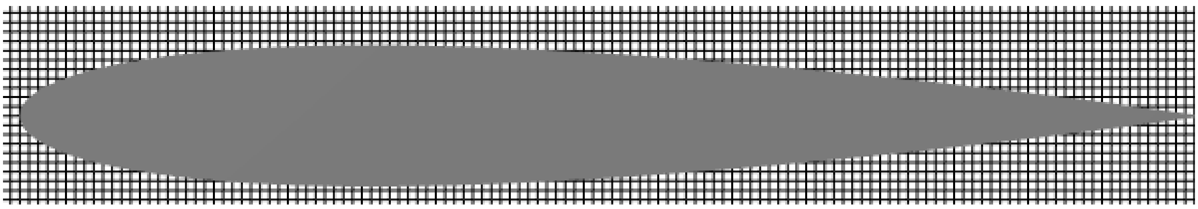
TurbEnergy(Phase #0) = **Pulsations**

Value = **0**

TurbDissipation(Phase #0) =**Turbulent scale**

Value = **0**

6.6.1.4 Initial grid



In the **Properties** window of the **Initial grid**, click the button to open the **Initial grid editor**.

Specify in the **Initial grid editor**:

for axis OX

Grid parameters

h_max = **0.634** [m]

h_min = **0.005** [m]

Insert a reference line with coordinate **x=0** [m].

Specify **Reference line parameters** for the reference line with coordinate **x=-2.54**:

h =**0.634** [m]

Specify **Reference line parameters** for the reference line with coordinate **x=0**:

h = **0.005** [m]

kh- = **1.001**

kh+ = **0.95**

Specify **Reference line parameters** for the reference line with coordinate **x=2.794**:

h	**=0.634**	[m]

for axis OY (click the button Y)

Grid parameters

h_max	**= 0.634**	[m]
h_min	**= 0.005**	[m]

Insert a reference line with coordinate **y=0** [m].

Specify **Reference line parameters** for the reference line with coordinate **y=-2.54**:

h	**=0.634**	[m]

Specify **Reference line parameters** for the reference line with coordinate **y=0**:

h	**= 0.005**	[m]
kh-	**= 1**	
kh+	**= 1**	

Specify **Reference line parameters** for the reference line with coordinate **y=2.54**:

h	**=0.634**	[m]

Click **OK** to close the **Initial grid editor** with saving the entered data.

Specify in properties of the **Initial grid**:

Grid structure = 2D

Plane = XY

In the **Properties** window of the **Initial grid** click **Apply**.

The main purpose of this example is to show the possibility of optimizing the airfoil's position of the profile by value of lift force. You may achieve a detailed solution by using a fine computation mesh. This example is simplified to increase speed of calculations. Therefore, the obtained results are not final for *NACA0012* airfoil. The statement of task requires finer computational mesh and decreasing the time step.

6.6.1.5 Parameters of calculation

Specify in the **Solver** tab:

- In the **Properties** window of the element **Time step**, specify:

Method	**= Via CFL number**	
Convective CFL	**= 50**	
Max step	**= 1**	[s]

- In the **Properties** window of the element **Stopping conditions > Time span**, specify:

Start at	**= 0**	[s]
Stop at	**= 0.05**	[s]

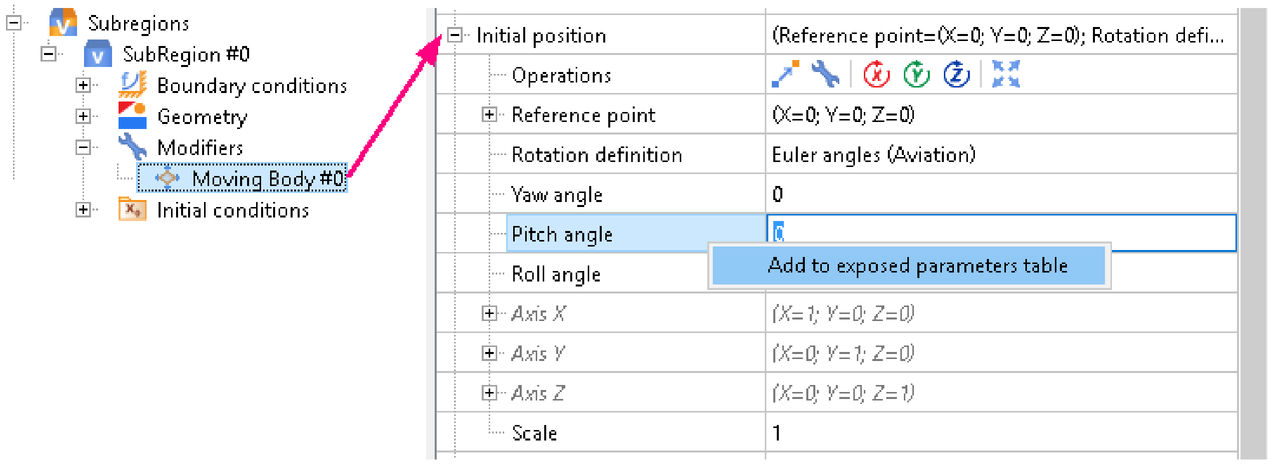
6.6.1.6 Optimization parameters

In this simulation, the optimization parameter is the angle between the profile and the stream. In order to make it changeable in *IOSO*, you have to define it as an external parameter.

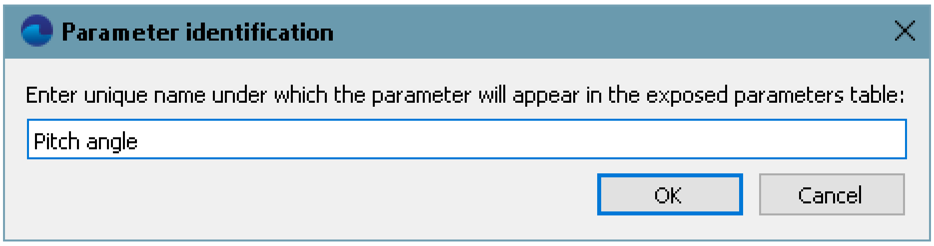
In order to make the pitch angle external parameter.

To define the attack angle as external parameter, do the following:

- From the context menu of the parameter **Initial position > Pitch angle** (which locates in the **Properties** window of **Moving body #0**) select **Add to exposed parameters table**:



- The **Parameter identification** window will open, enter **Pitch angle** there:



- After this you can disable updates of the **Moving body**. In the **Properties** window of the **Moving body**, specify:

Update

    Type **= Disabled**

6.6.1.7 Optimization criterion

In this problem, the optimization criterion is the lift force. Also, as additional information it is advisable to display the drag component force of the profile. To make these values be readable by *IOSO*, it is necessary to make them output parameters.

In the **Preprocessor** tab:

- Create **Characteristics** on **Imported object #0**.
- In the **Properties** window of **Characteristics #0**, specify:

Variable

    Variable **= Pressure**

In the **Solver** tab:

- From the context menu of the folder **Stopping conditions > User values**, select **Create**.
- In the **Properties** window of the new just created **Stop criterion #0**, specify:

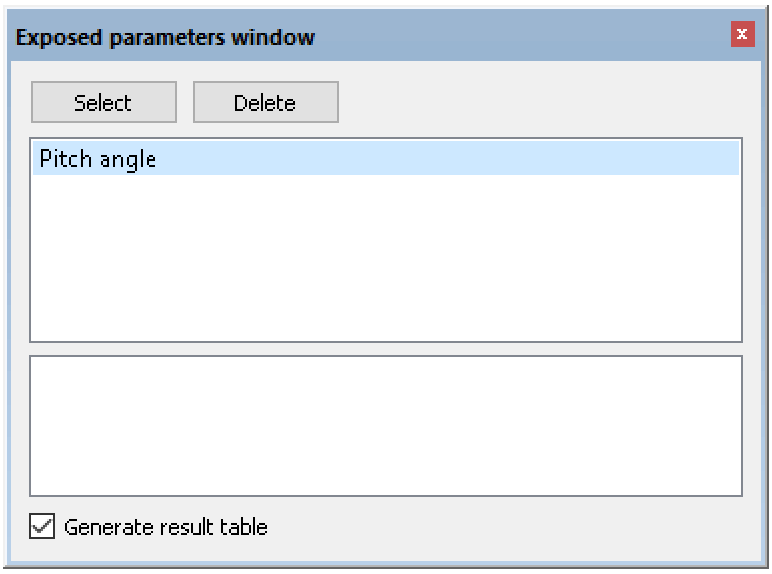
Name **= Fx**
Object **= Characteristics #0 (Imported object #0)**
Variable **= F fluid**
Component **= X**

- From the context menu of the folder **Stopping conditions > User values**, select **Create**.
- In the **Properties** window of the new just created **Stop criterion #0**, specify:

Name **= Fy**

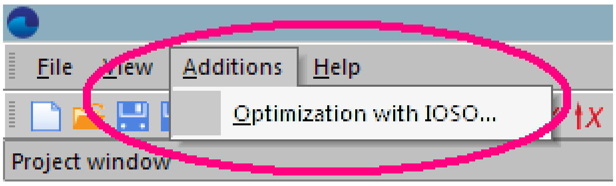
Object	**= Characteristics #0 (Imported object #0)**
Variable	**= F fluid**
Component	**= Y**

In the **Exposed parameters window**, select the **Generate result table** checkbox:

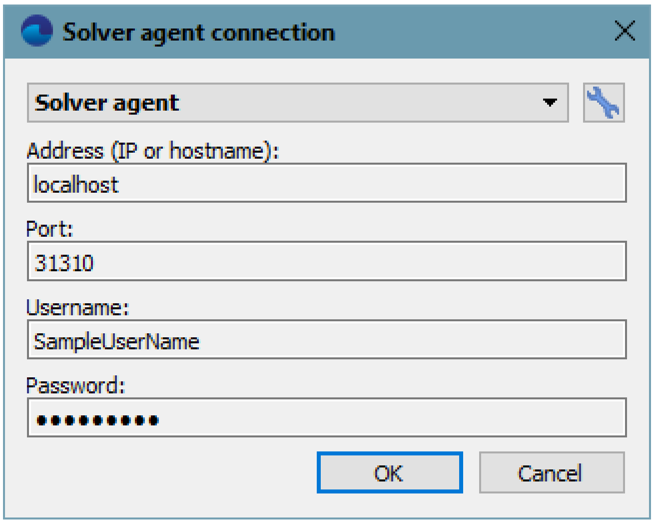


6.6.2 Setting up connection to IOSO

- Do not close the project, which has been created in *FlowVision*, and from the **Additions** menu select **Optimization with IOSO**:

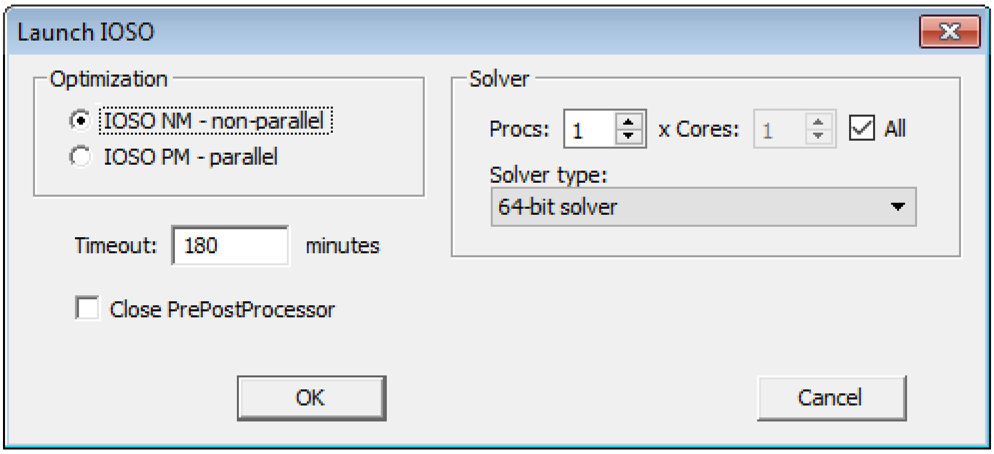


- Log on **Solver-Agent**:



- Specify the parameters of running the *IOSO*'s project and *FlowVision*'s solver:

Optimization	Select the version of *IOSO*, on which the computation will be done.
Timeout	Specify the maximal calculation time of one *FlowVision*'s project.
Solver	Specify the startup mode of the *FlowVision*'s solver.



The header of this dialog box contains the name and parameters of the used **Configuration** for connection to **Solver-Agent**.

- Click **OK**. After this the *FlowVision*'s project will automatically close and the *IOSO*'s project will open.

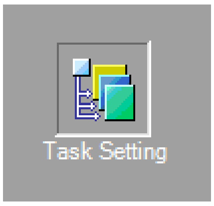
6.6.3 Preparing the project in IOSO

6.6.3.1 Optimization parameters

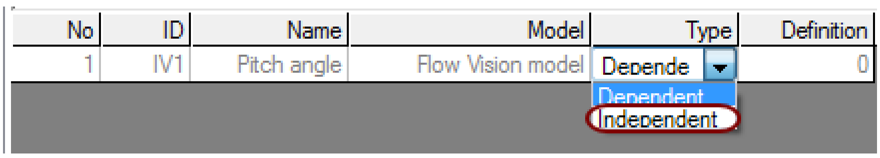
In the created project there is an already defined *FlowVision*'s mathematical model with input and output files. It is only required to set up the parameters and criteria for the optimization and their bounds.

Specify the optimization parameter and its limits of variations in *IOSO*:

- Switch to the **Task Setting** mode:



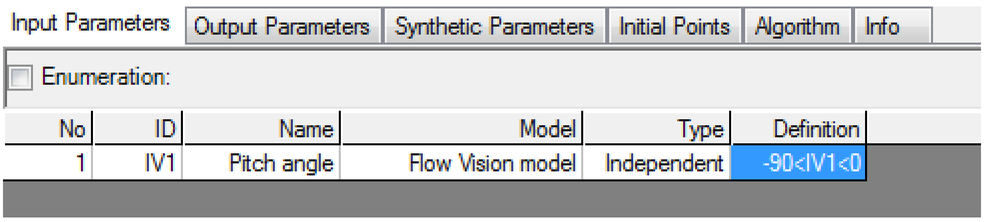
- In the **Input Parameters** tab, select **Pitch angle** and specify **Type =Independent**:



- Specify:

  Lower bound = -90

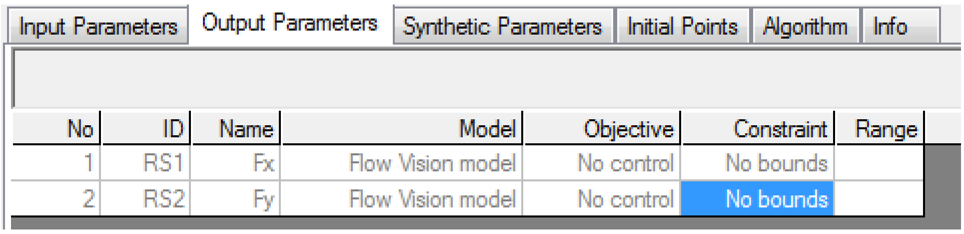
  Upper bound = 0



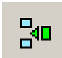
- Click **Ok**.

6.6.3.2 Optimization criteria

In *IOSO*, on the **Output parameters** tab, the parameters are displayed, which will be available for reading from *FlowVision* after the computation is finished:

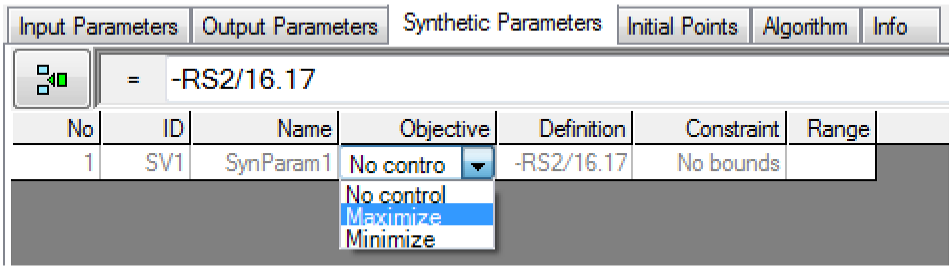


FlowVision calculates the forces acting on the flow by the profile; these forces are equal in magnitude and opposite in sign to the forces acting on the profile by the flow. In order to use the coefficient of the lift force (acting on the profile by the flow) as an optimization criterion, it is necessary to add a synthetic parameter and make it the optimization criterion:

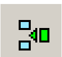
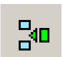
- In the **Synthetic Parameters** tab, click .
- Specify the formula for the new parameter as **-RS2/16.17** and click **OK***).
- Specify:

  Objective = **Maximize**

  Name = Cy



In this task, it is also advisable to display, as additional information, the drag coefficient and the aerodynamic efficiency ratio, which the ratio of lift force to drag force. To do this, you should create more synthetic parameters:

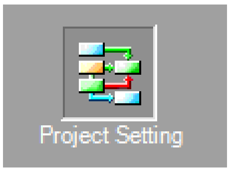
- Click .
- Specify the formula for the new parameter as **-RS1/16.17** and click **OK***).
- Specify this parameter's name as **Cx**.
- Click .
- Specify the formula for the new parameter as **RS2/RS1** and click **OK***).
- Specify this parameter's name as **k**.

Note:

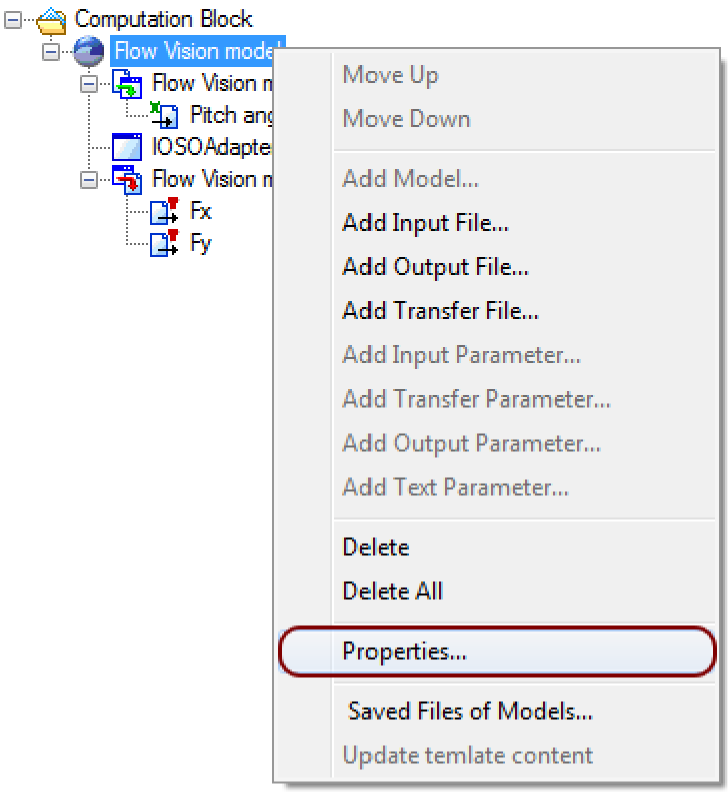
*) When you specify a synthetic parameter that depends on the output parameters, in the formula for the calculation, as the variables corresponding to the output parameters, their IDs are used, which are displayed on the **Output Parameters** tab.

6.6.3.3 Setting up computations

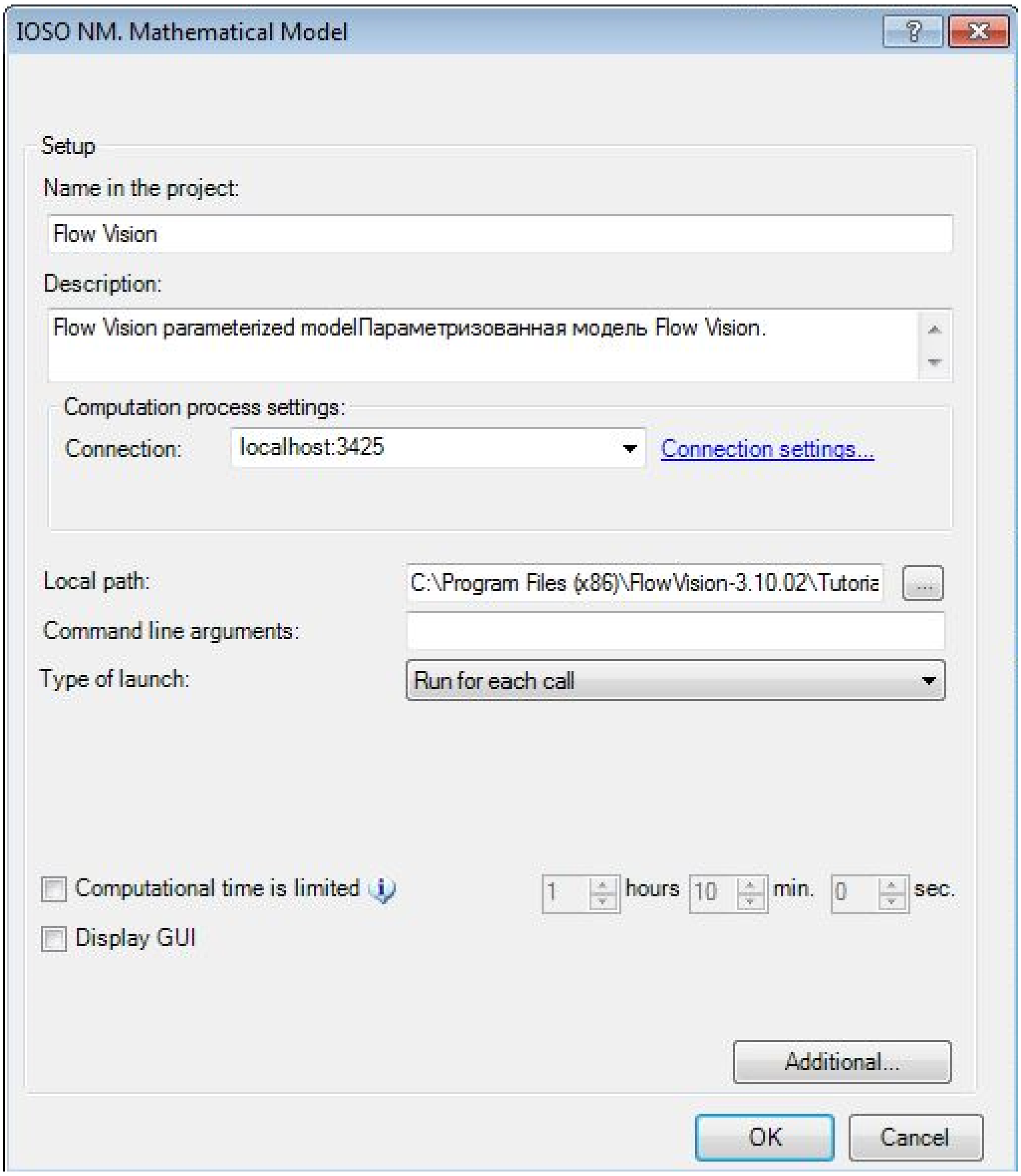
- Switch to the **Project Setting** mode:



- In the tree of the *IOSO*'s project, from the context menu of the item **FlowVision Model**, select **Properties**:

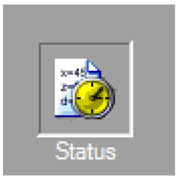


- In the **IOSO NM. Mathematical model** window, which opens, set the following tunings:
    - select **Run for each call** from the **Type of launch** drop-down list (this replaces the initial default option **Run for the first call only**)

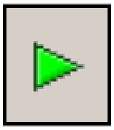


6.6.3.4 Running the optimization

- Switch to the **Status** mode:



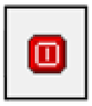
- To start the optimization, click the button:



- After this, an the time when the of the *FlowVision*'s projects were started for the computation and elapsed time of these computations will be indicated:

Model	Status	Computation time limit	Current computation ti...	Previous computation time	Total computation time
Flow Vision	Running		00:00:08	00:00:00	00:00:08

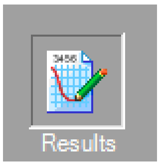
- Optimization can be stopped by clicking the button:



It is recommended to stop the optimization when the optimization results do not change much.

6.6.3.5 Viewung results

When about 10 computations are done, viewing the results will be available. To display the results of the computation, switch to the **Results** mode and open the **Search History** tab.



In this tab, the results of all *FlowVision*'s computations are displayed:

Objective Improvement | Search History | Info

Call No	Pitch angle	Cy	Cx	k
22	-33.7989580222	**1.5307836524**	1.0099027803	0.6597292692
23	-44.2925249424	**1.5140370190**	1.6494371239	1.0894298510
24	-55.7822340055	**1.9900819828**	2.9719073313	1.4933592470
25	-56.0226699748	**2.0772520082**	3.1701295157	1.5261169579
26	-57.9989611996	**2.0038055441**	3.3152914377	1.6544975871
27	-51.9540916233	**1.6504866663**	2.4445108787	1.4810848998
28	-54.1125536812	**1.8609365514**	2.7613868293	1.4838694136
29	-57.1850618429	**1.8568100268**	3.0670134808	1.6517648206
30	-54.4651493383	**1.7596239698**	2.6843295864	1.5255131963
31	-53.3620598508	**1.6142394273**	2.3690866446	1.4676178790
32	-56.8049150080	**1.8252930581**	2.9330192350	1.6068757957
33	-60.1868522780	**1.6557007923**	3.0501867622	1.8422330752
34	-57.0952897826	**1.8536597131**	2.9564689641	1.5949361920
35	-55.2544246934	**1.7713107134**	2.8121750691	1.5876238132
36	-56.4873579636	**1.9484184896**	3.0983379348	1.5901809346
37	-57.3276480319	**1.5706477712**	2.7656852007	1.7608564131
38	-56.3915441942	**1.9941684928**	3.0711158449	1.5400483238
39	-54.3364088956	**1.7514999282**	2.6142920249	1.4926018453
40	-55.3507309584	**1.7062220449**	2.7411835253	1.6065807691

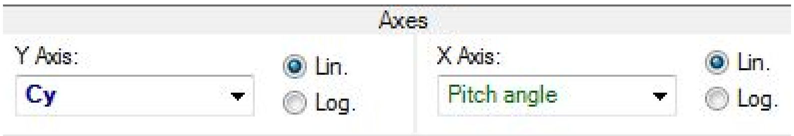
To display a plot with results of the optimization, do the following:

- At the bottom of the table, select **Type of displaying** as **Table+Plot** (click the appropriate button):

- Above the plot, which appears, click the **Plot** button ().

- Specify:

X Axis	**= Pitch angle**
Y Axis	**= Cy**



- In the settings of displaying the result select **Filtered only**:

- In the *IOSO* window, a plot of the dependency of **Cy** on the pitch angle (the angle of attack) will be displayed: