

QUICK START

1. Click **Open Project** and select VisiMix file with extension **.vx3**
2. To stop simulation select **Reset** button
3. To restart simulation select **Run**
4. To change points of tracer inlets select **Change inlet Positions**

VISIMIX 3D Viewer. User's Guide

VisiMix 3D Viewer provides 3-dimensional visualization of macro-scale mixing based on results of VisiMix Turbulent simulation of average velocity field and characteristics of turbulence for the corresponding project. The program demonstrates course of single phase mixing by following trajectories of particles and dispersion of a cloud of injected tracer particles in a real time, accelerated and slow regimes. It allows for observation of mixing in the tank from different positions and angles.

Simulation of mixing consists in reproduction of movement of colored balls and of dispersion of clouds of small particles in agitated media under effect of circulation and macro-scale turbulence. It is supposed that density of the balls and particles does not differ from density of liquid media. They are injected with initial velocity 0 (zero).

1. Preparation of VisiMix project for 3D simulation.

To apply the 3D Viewer, open the selected VisiMix project in the VisiMix Turbulent program (Fig 1) and perform modeling corresponding to functionality of the **Hydrodynamics** and **Turbulence** sections of the **Calculate** menu.

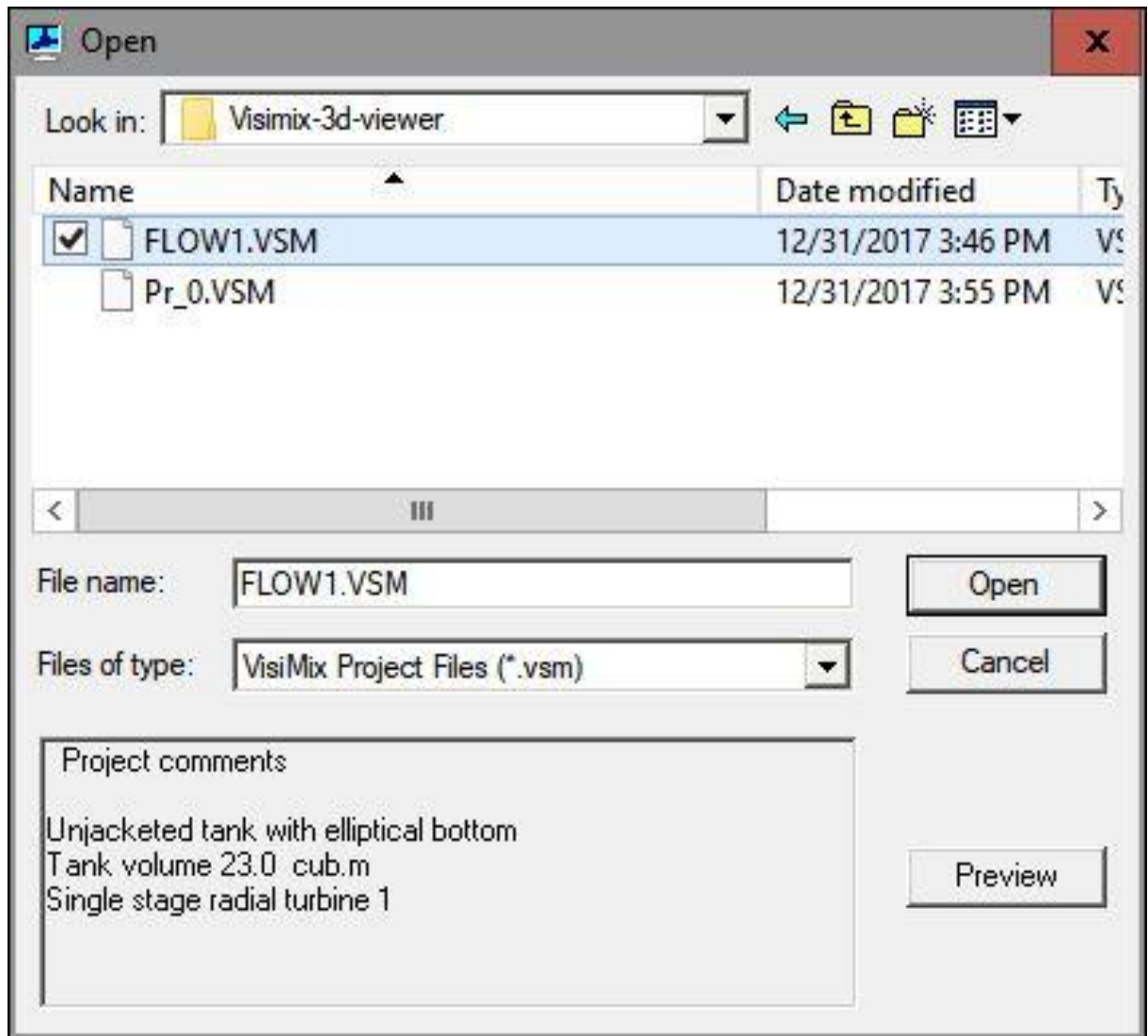


Figure 1.

Select **3D VISUALISATION** in the Main Visimix menu section (Fig.2) and save results of modeling as a file with extension ***.vx3** (Fig.3).

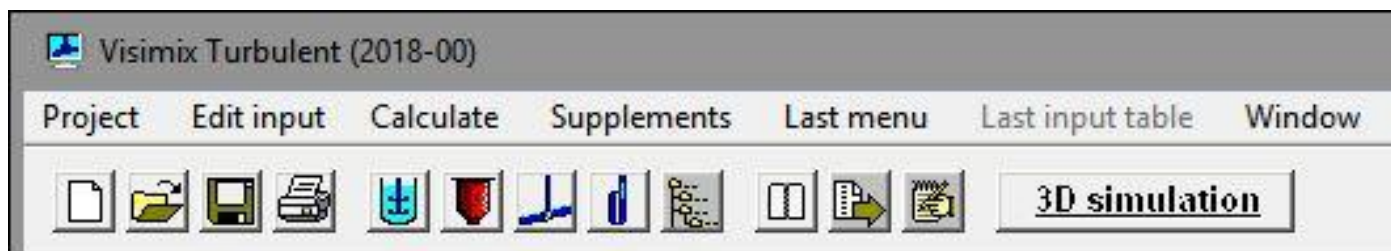


Figure 2.

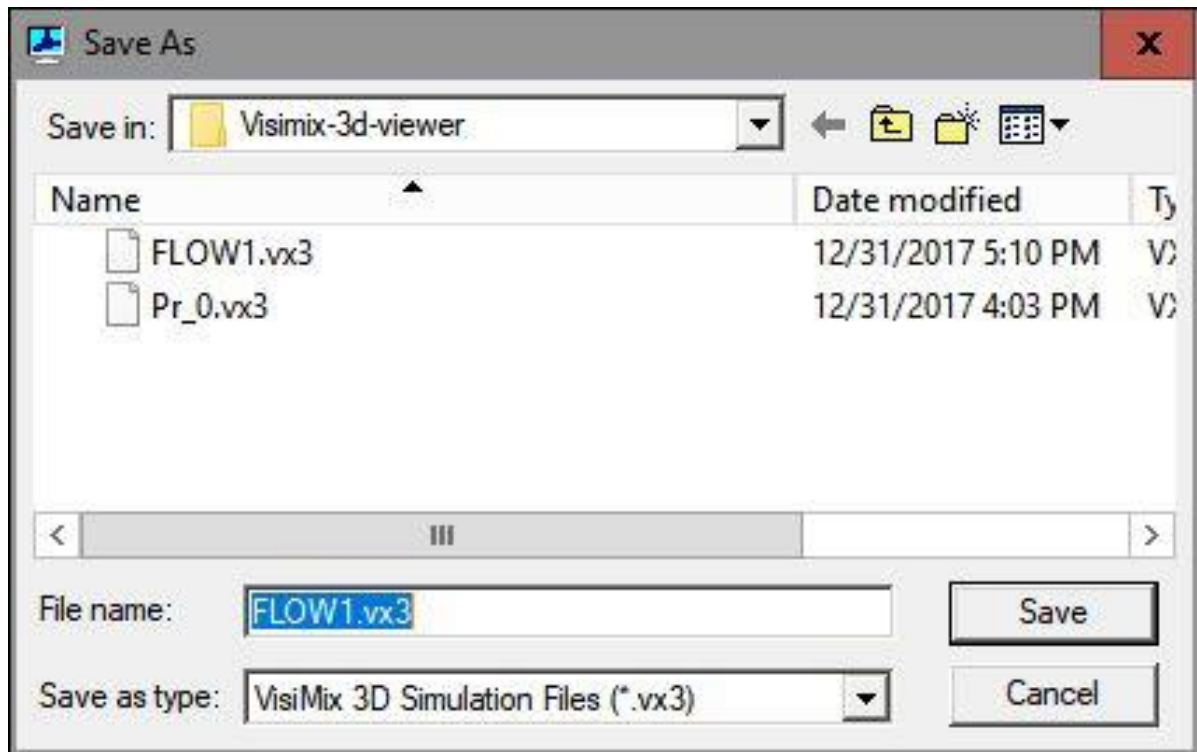


Figure 3.

You can also create the ***.vx3** file using the menu option *Project >3D SIMULATION* (Fig.4).

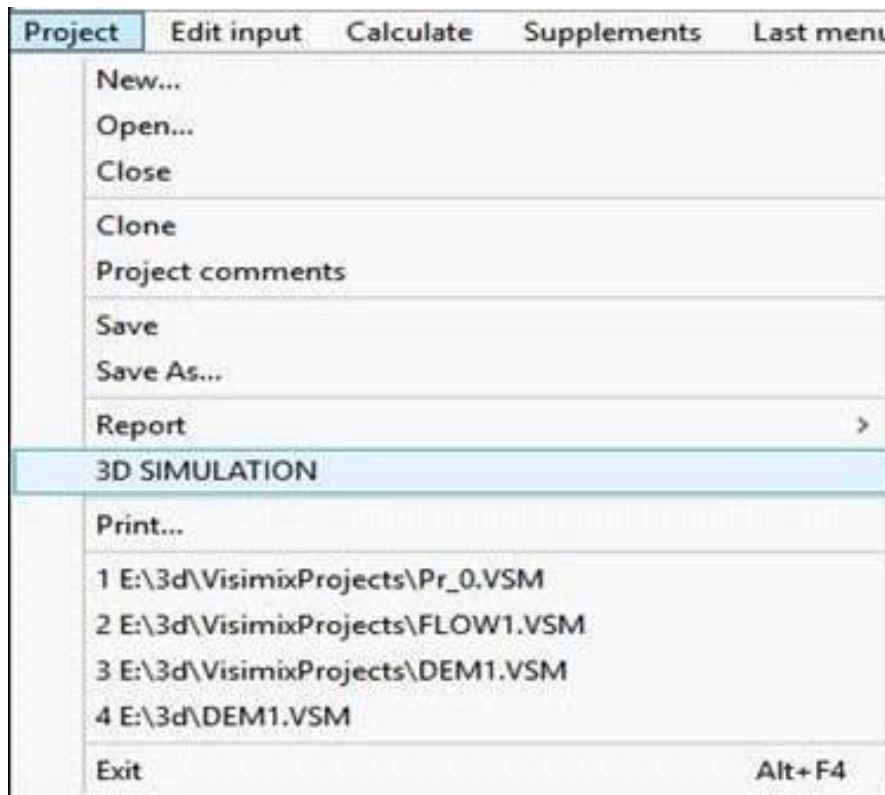


Figure 4.

1. Starting of the VISIMIX 3D VIEWER

Open program **Visimix 3D Viewer**. You receive the following window:

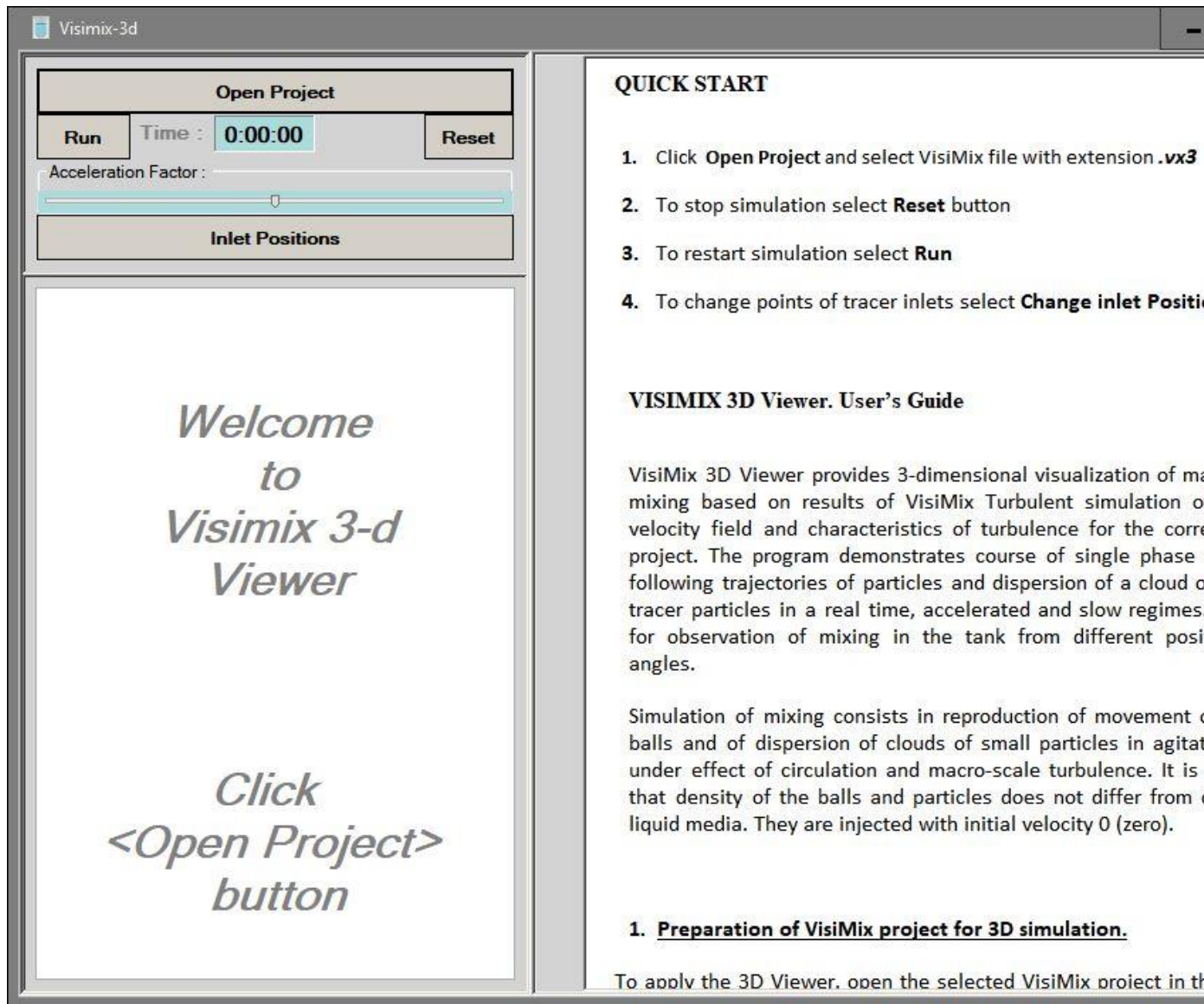


Figure 5.

Click the **Open Project** button and open the ***.vx3** project file in corresponding directory (Fig.6).

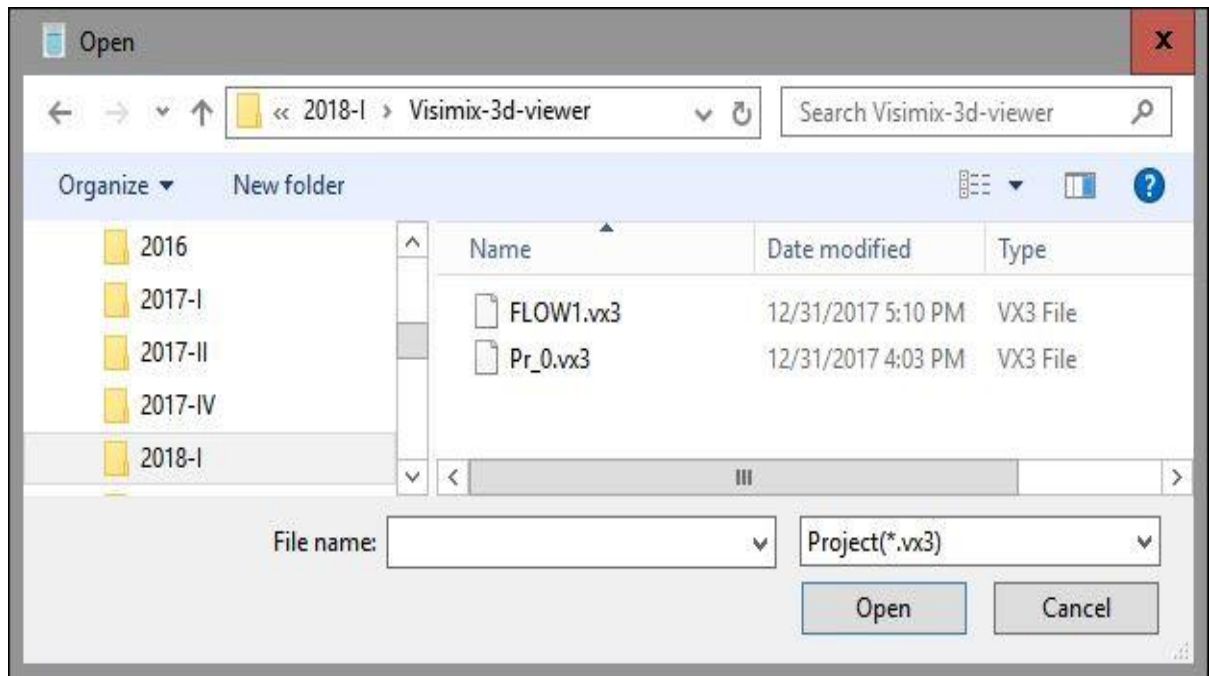


Figure 6.

The program returns **Simulation panel** with 3D image of mixing in the right part of the screen and control and information section in the left part (Fig.7)

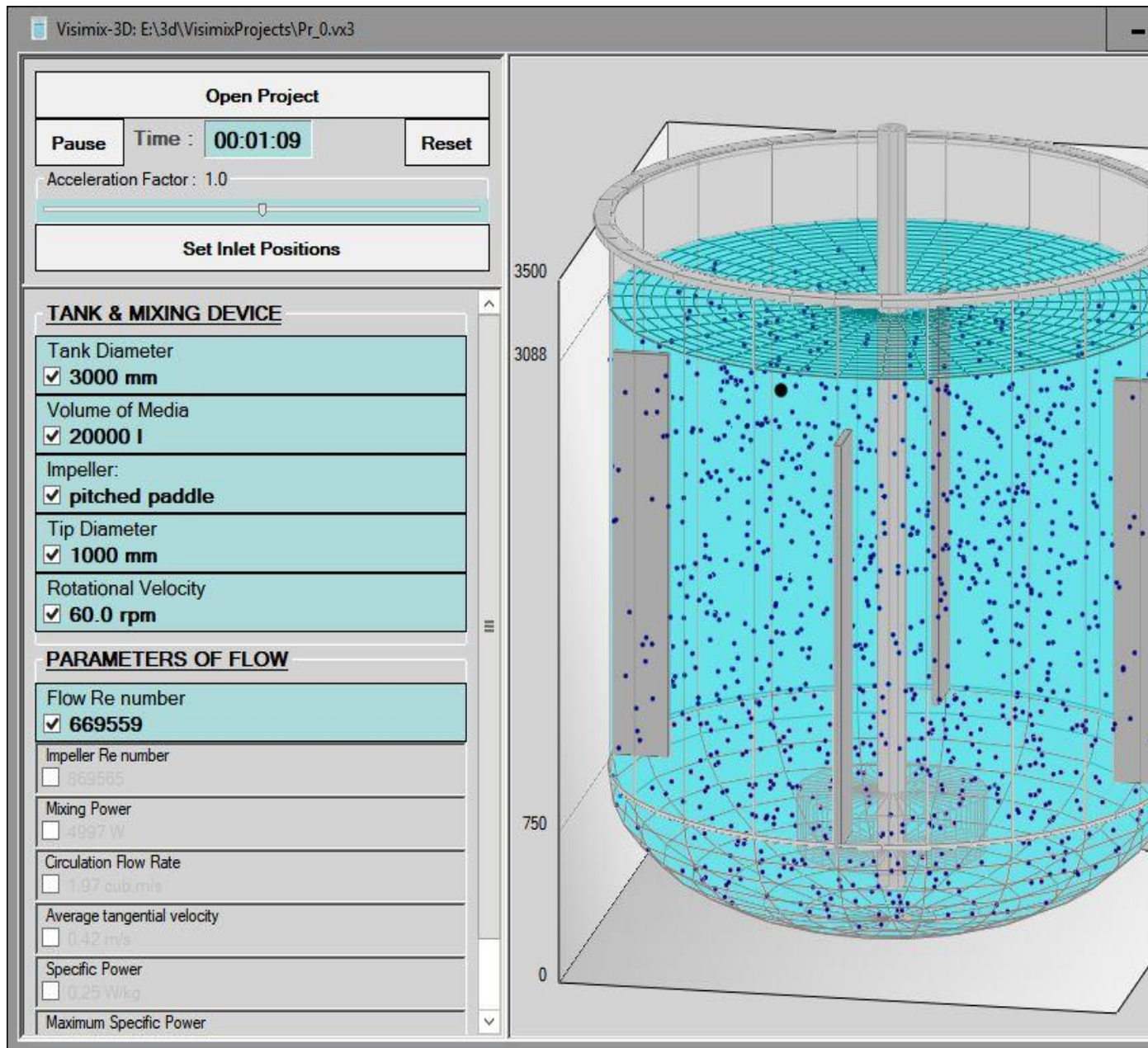


Figure 7.

3. Simulation panel

Open Project. This button is used to open new or replace the current 3D project file with another one.

Run. This button restart simulation after it was stopped by clicking the bottom ***Pause*** and starts the mixing simulation by injection of

tracing balls and particles. In this case the Run button simultaneously activates the timer in the **Time** box. After tracing has been started, the **Run** button is replaced with the button **Pause** (fig.7). Pushing this button interrupts mixing, stops the timer in **Time** box and restores the **Run** option (Fig.8).

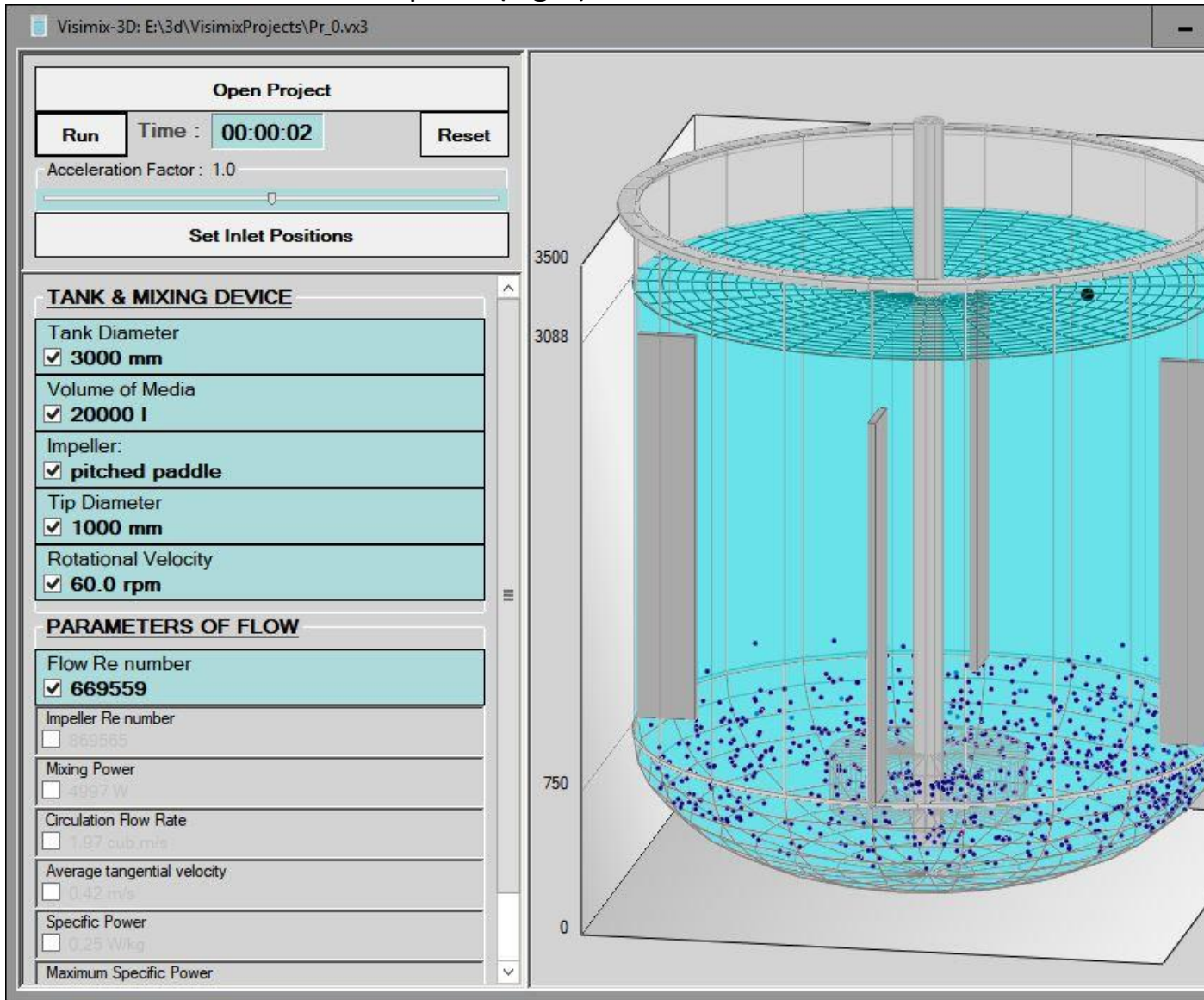


Figure 8.

Reset. This button is used to stop mixing and return the system to initial state before injection of tracers - colored balls and clouds

of particles. In this case the **Run** buttons starts simulation from beginning.

The **Acceleration Factor slicer** serves for changing **time scale**, it allows to decelerate or accelerate visual imaging by factor shown in the screen. Figures in the **Time** information box shows real time from the start of mixing and are not connected with the **time scale of** imaging.

Change inlets position

This button opens the *Control panel* (Fig. 13) that contains tools for selection of starting positions for balls and injection points of clouds of particles.

Reference information. Lower part of the left section provides information on main characteristics of tank and mixing device and also - some results of mathematical modeling related to flow velocities and turbulence in the tank. To bring up values of the reference parameters, mark the corresponding checkboxes (Fig.9).

Open Project		
Run	Time : 00:14:04	Reset
Acceleration Factor : 1.0		
Change Inlet Positions		
<u>TANK & MIXING DEVICE</u>		
Tank Diameter		
<input checked="" type="checkbox"/> 3000 mm		
Volume of Media		
<input checked="" type="checkbox"/> 20000 l		
Impeller:		
<input checked="" type="checkbox"/> pitched paddle		
Tip Diameter		
<input checked="" type="checkbox"/> 1000 mm		
Rotational Velocity		
<input checked="" type="checkbox"/> 60.0 rpm		
<u>PARAMETERS OF FLOW</u>		
Flow Re number		
<input checked="" type="checkbox"/> 669559		
Impeller Re number		
<input type="checkbox"/> 369565		
Mixing Power		
<input checked="" type="checkbox"/> 4997 W		
Circulation Flow Rate		
<input type="checkbox"/> 1.97 cub.m/s		

Figure 9.

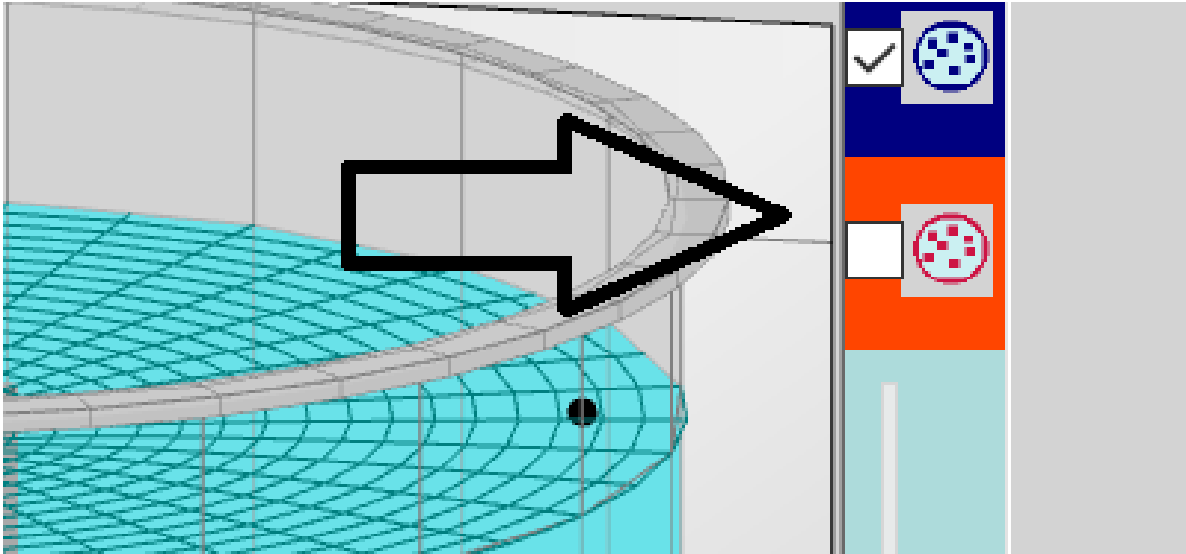


Figure 10.

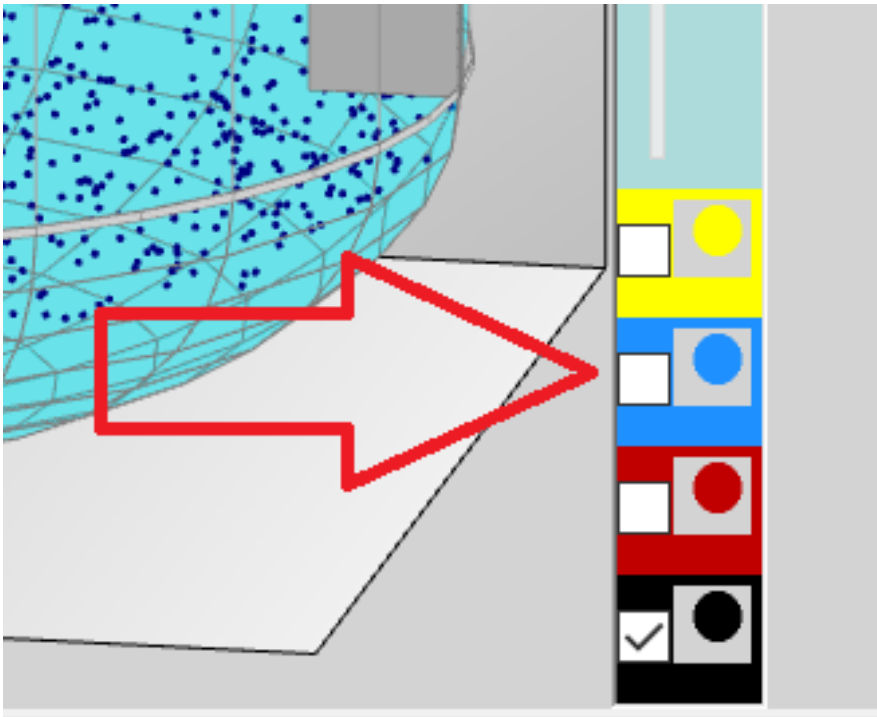


Figure 11.

Upper check boxes (Fig. 10). Marking the check boxes in the upper right part of the simulation panel activates corresponding cloud of particles.

Lower check boxes (Fig. 11). Marking the check boxes in the lower right part of the simulation panel activates corresponding colored balls.

Vertical slider (Fig.12) allows to incline the tank and observe mixing from different angles.

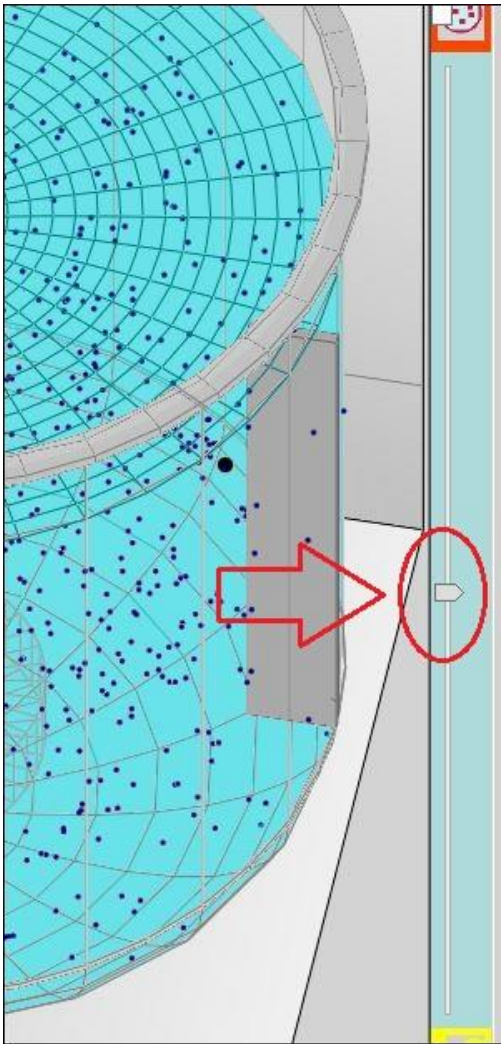


Figure 12.

5. Control panel.

This panel (Fig.13) serves for selection and changing of initial position of clouds and colored balls before beginning of simulation.

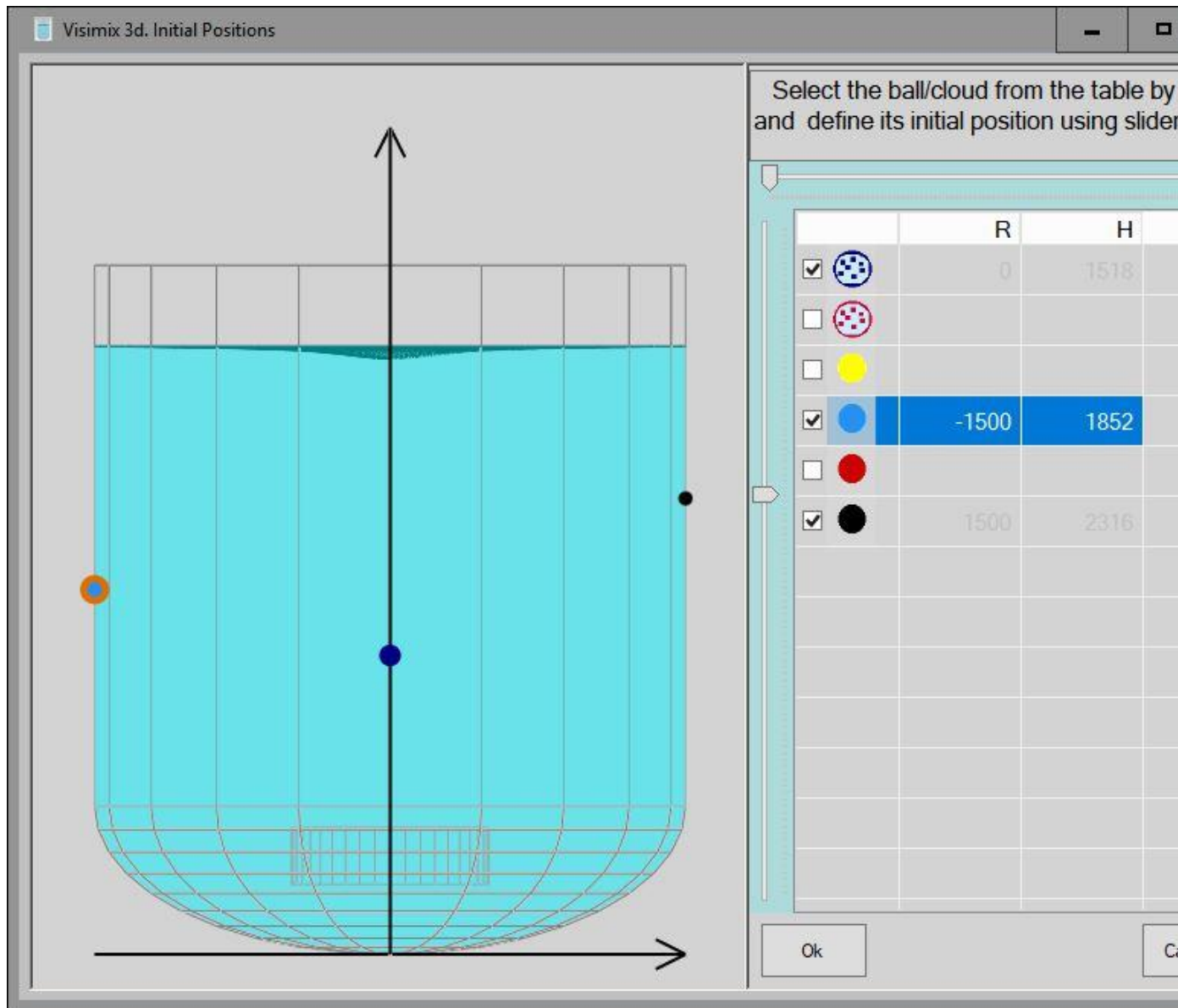


Figure 13.

Marking the checkboxes in the right part of the **Control panel** activates selected clouds of particles and colored balls. Different clouds and balls are independent and can be activated separately or simultaneously. The activation is duplicated in the corresponding checkboxes of the **Simulation panel**.

Initial positions of the selected clouds and particles are fixed or changed using the **vertical and horizontal rulers** Radial and axial coordinates with respect to axis and bottom of the tank arrive in the table. Corresponding positions in the tank shown on the scheme in the left part of the panel.

In order to start simulation, close the Control panel with **OK** and click **Run** in the **Simulation panel**.