VisiMix® OFF-CENTER
Quick Start Manual and an Application Example

1. Introduction

VisiMix OFF-CENTER is a brand-new software tool for mathematical modeling and design of mixing tanks with one or two identical or different impellers installed at an **off-centered shaft**.

The projects created and saved with VisiMix OFF-CENTER tool have extension .vxo.

Terms **nonconcentric shaft** or **off-center shaft** mean that the shaft axis does not coincide with the vertical axis of the tank.

This tool is a complimentary software incorporated into VisiMix Turbulent.

This version, which is the first one for the tool, includes an analysis sections covering simulation of hydrodynamics, turbulence, single-phase liquid mixing and heat transfer phenomena in cylindrical tanks with one or two impellers on a shaft of any configuration selected from the following list:

- a vertical off-center shaft;
- an inclined shaft; or
- a bottom-entering shaft.

The program supports configurations with one or two impellers installed on the same shaft. It enables user to conduct analysis of hydrodynamic characteristics of each impeller with consideration for interaction of the impellers with each other. Asymmetric position of the mixing hardware results in highly complicated flow pattern. Therefore, simulation is based on a simplified description of this pattern that is still sufficient to enable engineers to design mixing tanks and reactors with off-center shafts and analyze characteristics of processes conducted there.

2. User’s Guide and an Application Example for a New Project

Step 1. Open VisiMix OFF-CENTER

Open VisiMix OFF-CENTER tool. The main toolbar shown below is displayed on the screen:

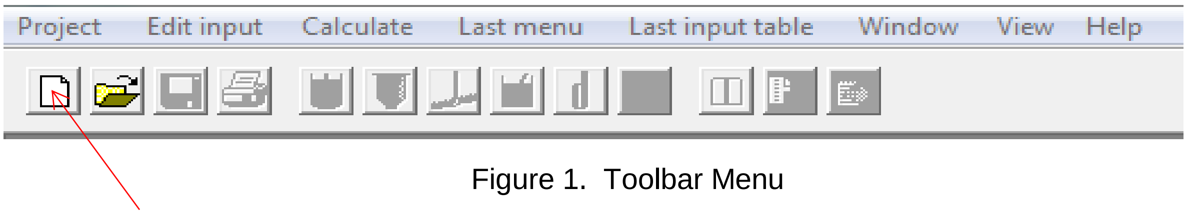


Figure 1. Toolbar Menu

Project | New

To start a new project either click the new project quick-start button designated in Fig. 1 OR:

- Select **New** under the **Project** tab.
- When the new project dialogue appears, enter a project name in the name field, then click Save.

Step 2. Select Tank Configuration

Entering input data for a new project | Tank

After you enter the name of a new project and click OK, a selection of tank configurations (Fig. 2) is displayed. Each configuration is characterized by its own types of the bottom (flat, conic or elliptical) and the heat transfer hardware (insulation without a jacket, or conventional, half-pipe coil, or embossed/dimpled jacket).

The jacket may comprise one or two sections connected in series or in parallel. Choose a tank by clicking anywhere inside the selected sketch. The tank selected is displayed in the **Current choice** window on the right. Click OK to confirm your choice.

NOTE:

If you do not expect to conduct heat transfer analysis for the current project, select an unjacketed tank from the **Insulated Tanks without Jacket** group rather than a jacketed tank, even if your tank is outfitted with heat transfer hardware.

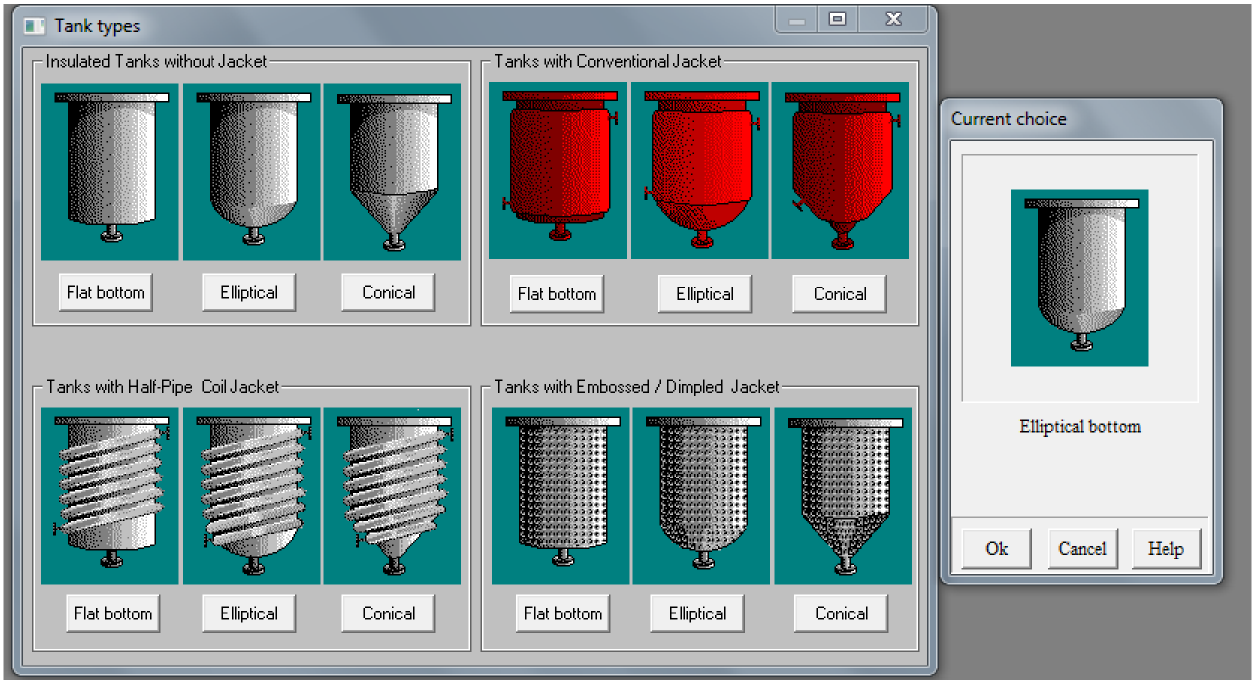


Figure 2. Tank Type Selection

When the sketch of a selected tank is displayed in the **Current choice** box at the right, enter the requested values of tank parameters by completing the table of dimensions.

We continue discussion of this example by selecting **Insulated Tanks without Jacket** with **Elliptical** bottom. Press **OK**. The input window for tank geometry is displayed, see Fig. 3.

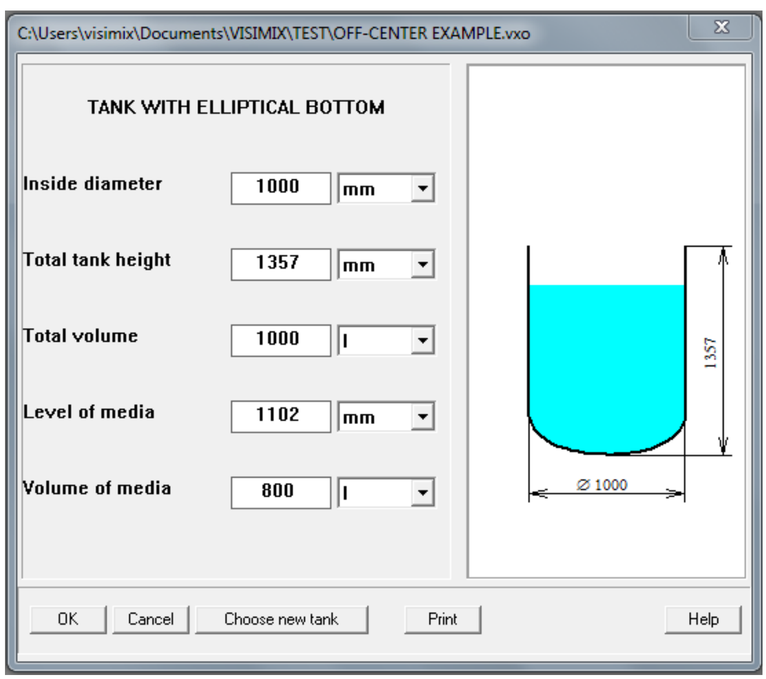


Figure 3. Tank Geometry Input Window

Enter the three input tank characteristics: Inside diameter, Total tank height (or Total volume) and Level of media (or Volume of media).

NOTE:

The **Total tank height** is a distance from the lowest point of the bottom to the upper edge of the joint between the tank and its head.

Press **OK**.

Step 3. Select Baffle Configuration

The baffle type selection window is displayed as shown in Fig. 4. In this example, we select the **No Baffle** option.

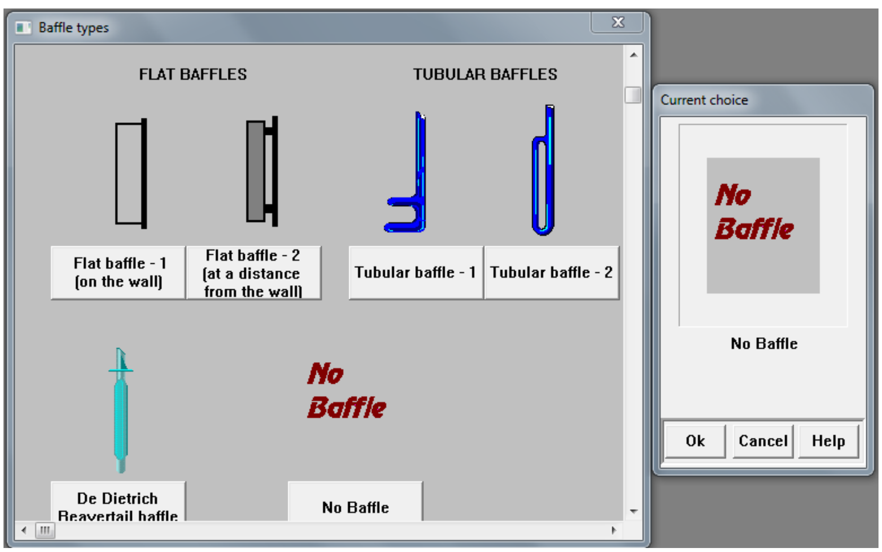


Figure 4. Baffle Type Selection

Press **OK**.

Step 4. Select Impeller Configuration

The **Impeller Design** window is displayed as shown in Fig. 5.

Enter **Rotational speed** and **Motor Power**. By pressing **Add**, user adds an impeller to the shaft. The impeller type selection window is displayed as shown in Fig. 6.

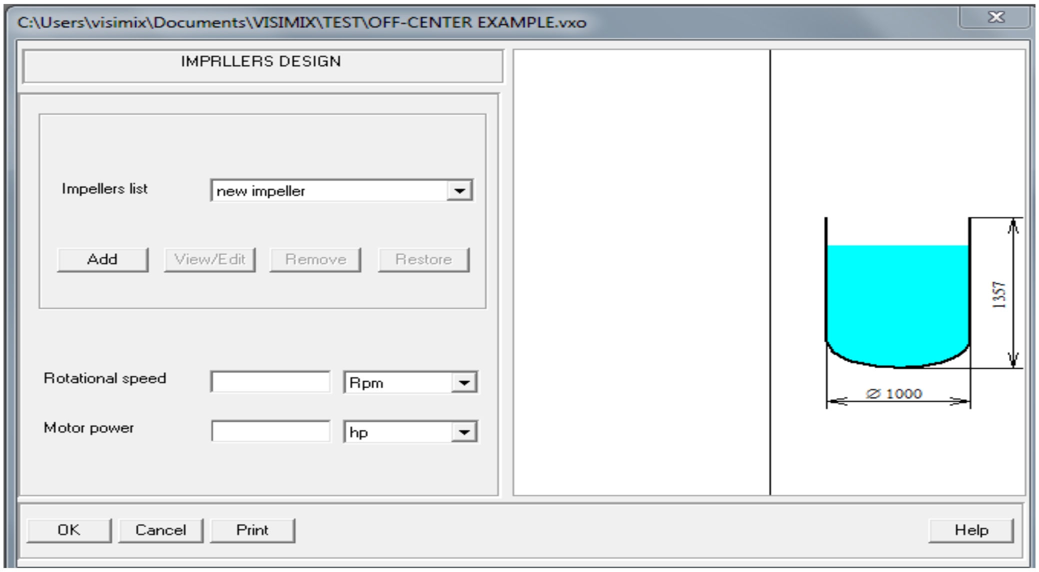


Figure 5. Impeller Design Window

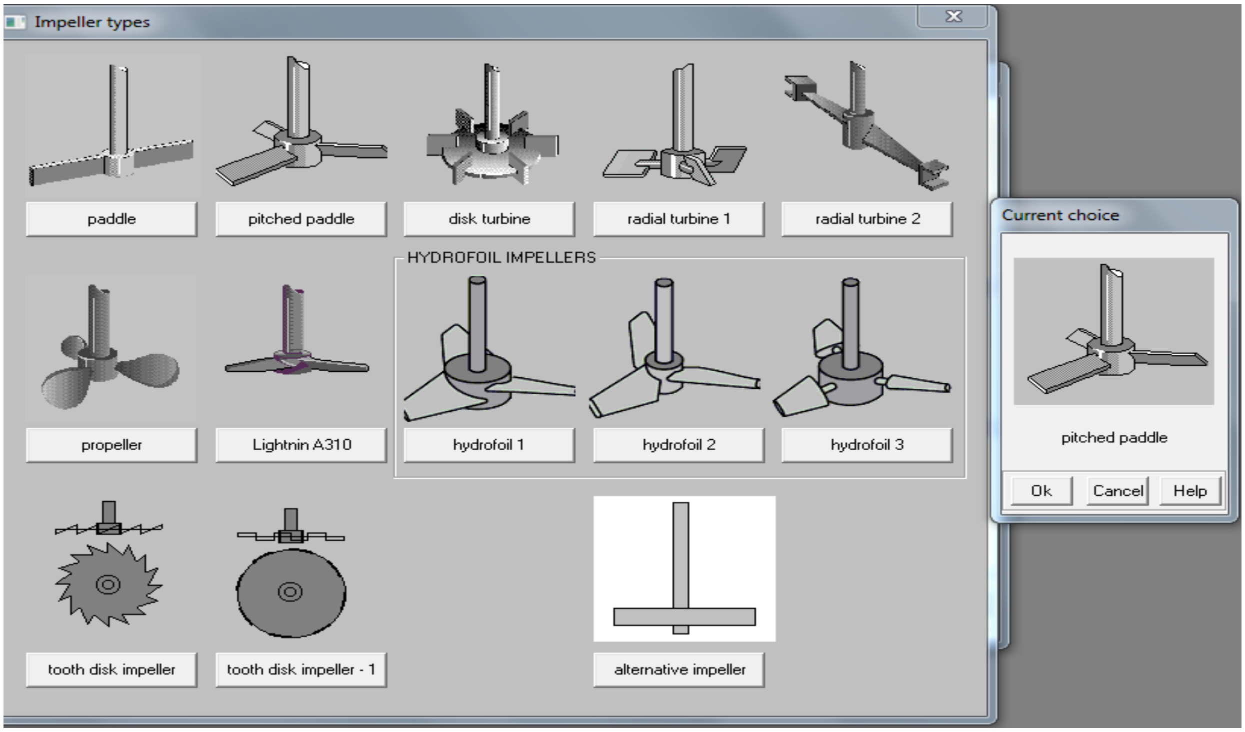


Figure 6. Impeller Type Selection Window

Select an impeller type. In this example, we select **pitched paddle** and press **OK**.

Step 5. Select Impeller Dimensions and Add a Second (Optional) Impeller

The impeller geometry input window is displayed, see Fig. 7. Enter impeller characteristics and press **OK**. The same impeller dimension window with all parameters entered is displayed as shown in Fig. 8.

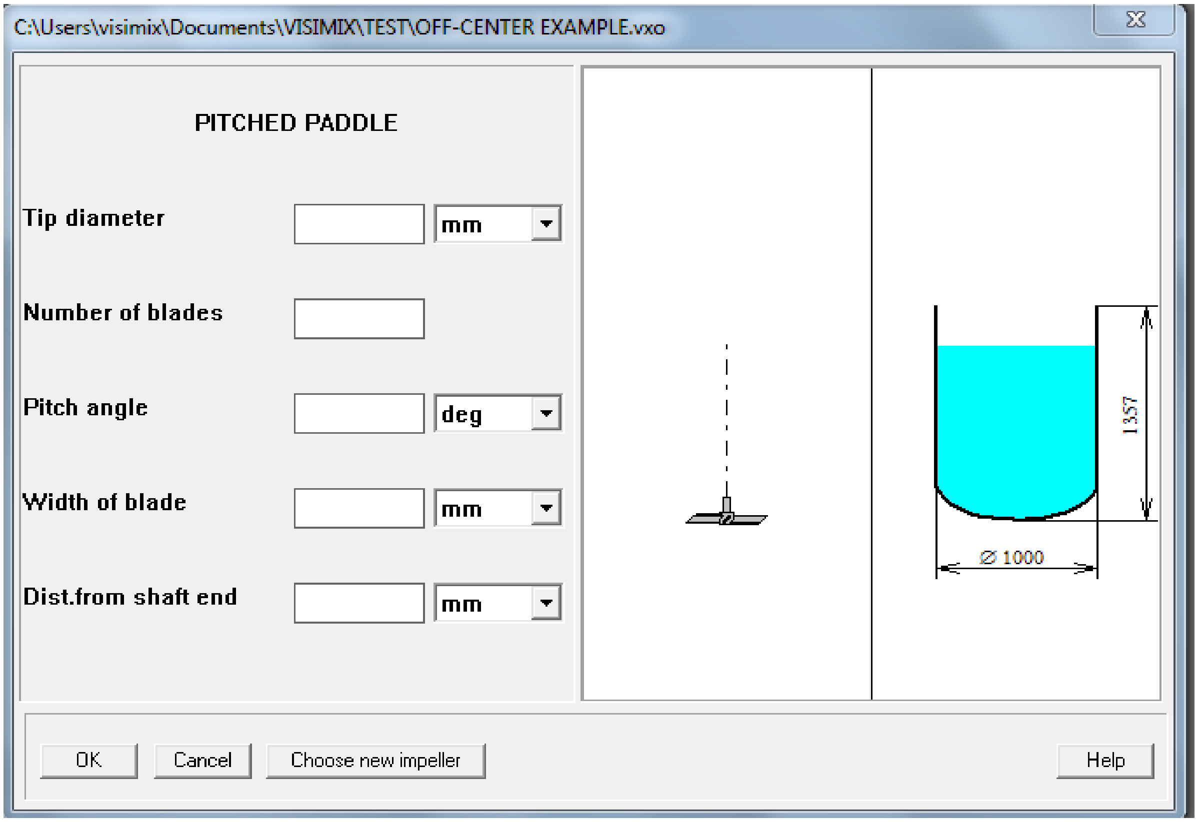


Figure 7. Impeller Geometry Input Window

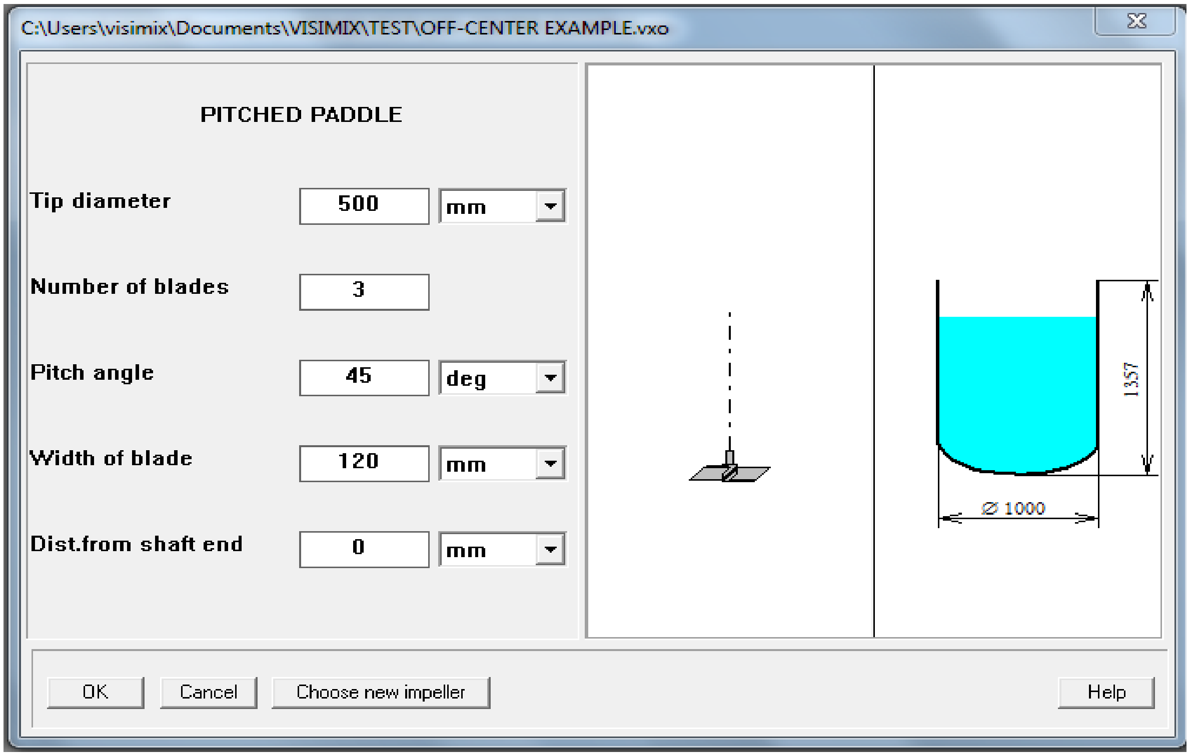


Figure 8. Selected Impeller with Dimensions

Press **OK**, and a new window shown in Fig. 9 is displayed. “!1” shown in red at the left from the impeller sketch indicates that the first impeller is selected.

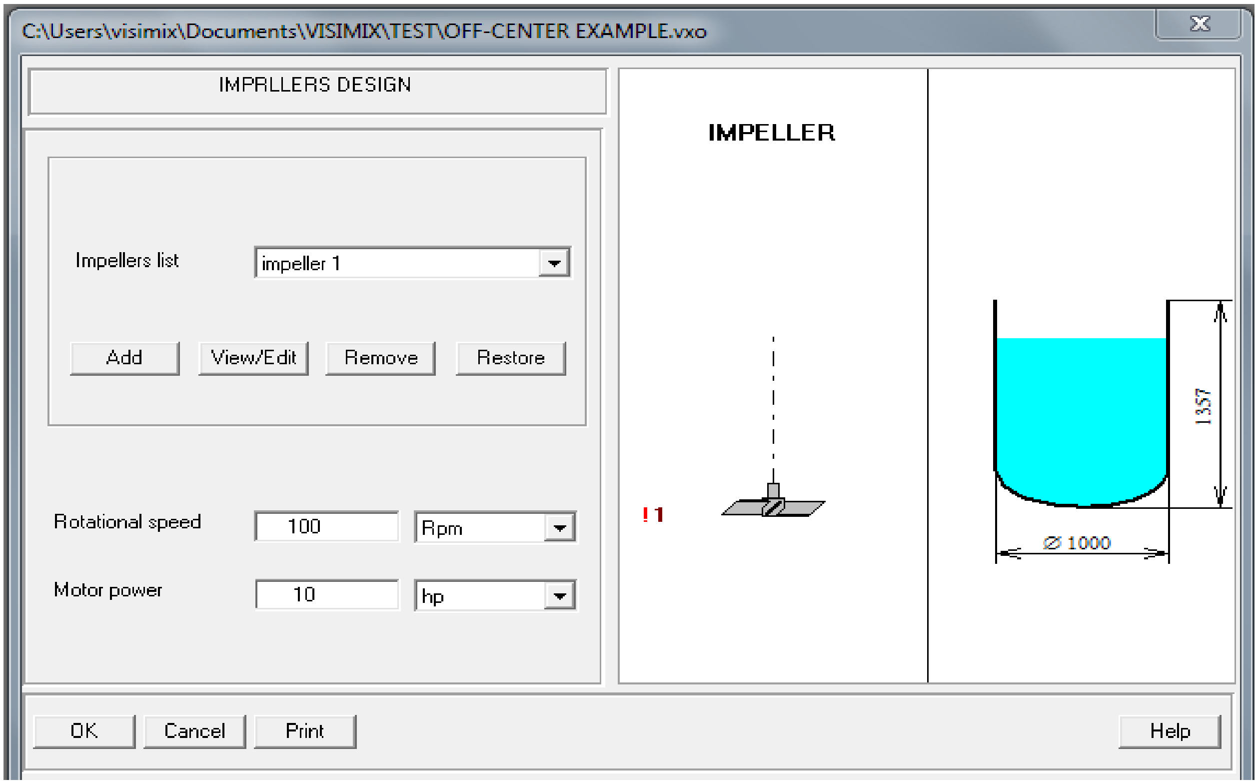


Figure 9. Impeller Design Window after Data Input

If the reactor has two impellers, press Add button shown in Fig. 9 to add the second impeller. Then, select a type of the second impeller per Fig. 6 and its geometry per Figs. 7 and 8.

After clicking OK in Fig. 8 for the second impeller, a new window with design of a two-level impeller is displayed as shown in Fig. 10. The impeller sketch is changed, and “!2” is displayed now at the left from the second impeller indicating that this impeller is currently selected.

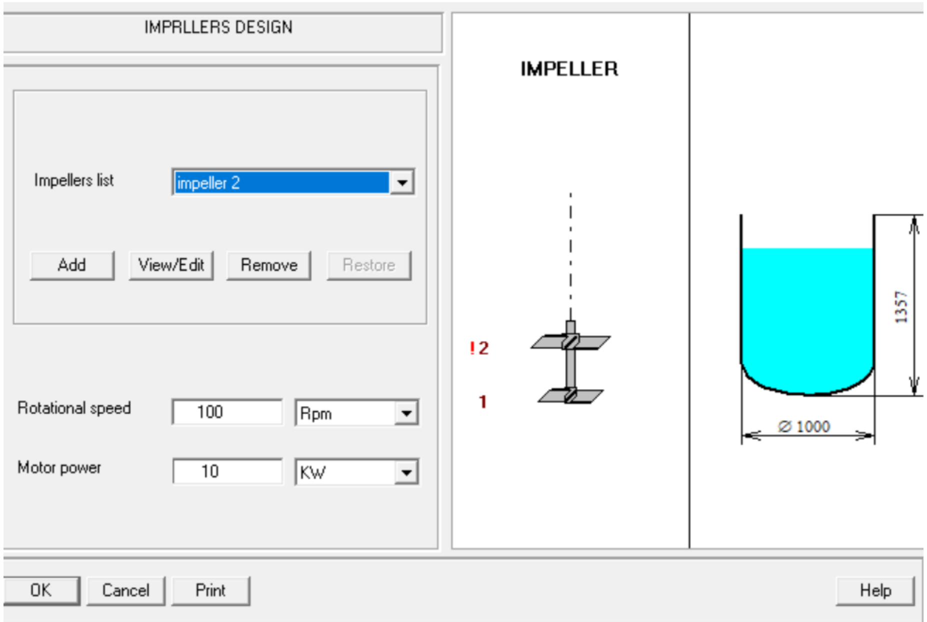


Figure 10. Impeller Design Window after Input of the Second Impeller Data

NOTE:

For the first impeller we entered zero as a distance from the shaft end (see Fig. 8). For the second impeller, this distance should be positive.

It is possible now to select either impeller by clicking it in Fig. 10 and edit parameters of the selected impeller or remove it.

In this example, a configuration with a single impeller is used.

Select impeller 2 and press **Remove**. A configuration with one impeller is displayed as shown in Fig. 11.

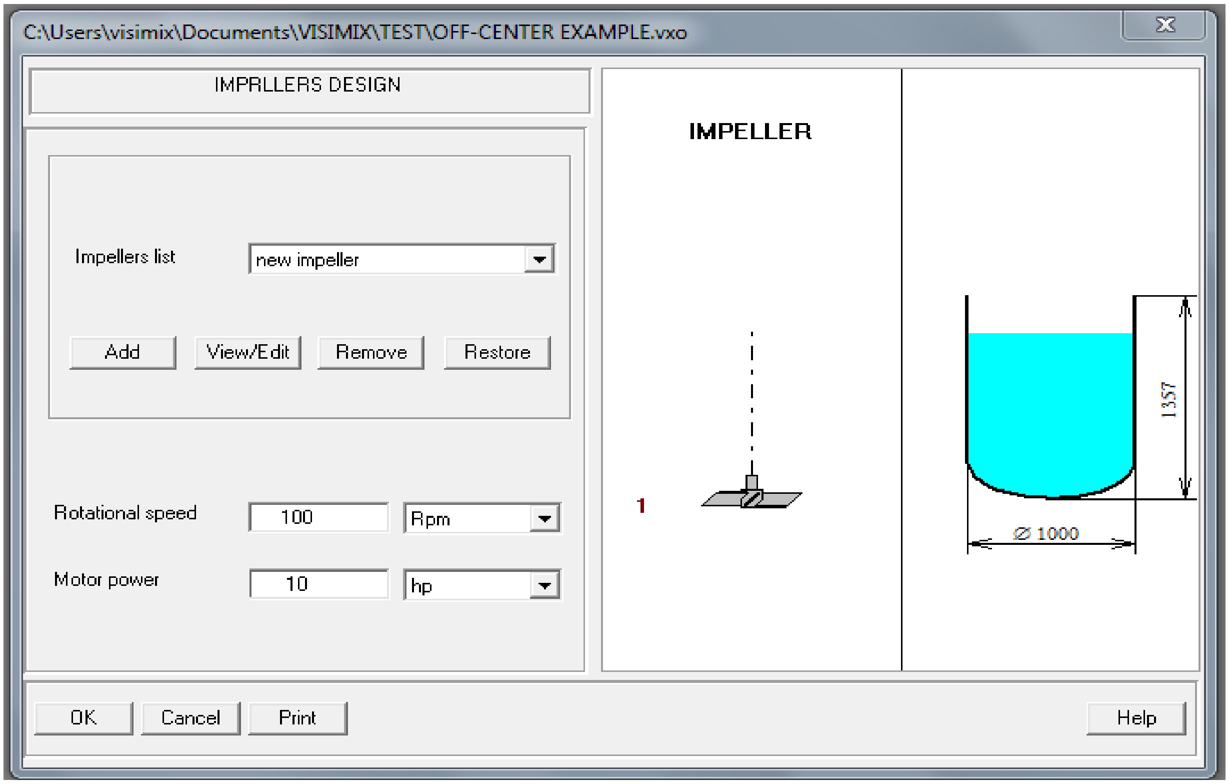


Figure 11. Impeller Design Window. One-Impeller Configuration

Click **OK**. A new window, **Shaft Position Options**, is displayed as shown in Fig. 12.

Step 6. Select Shaft Configuration

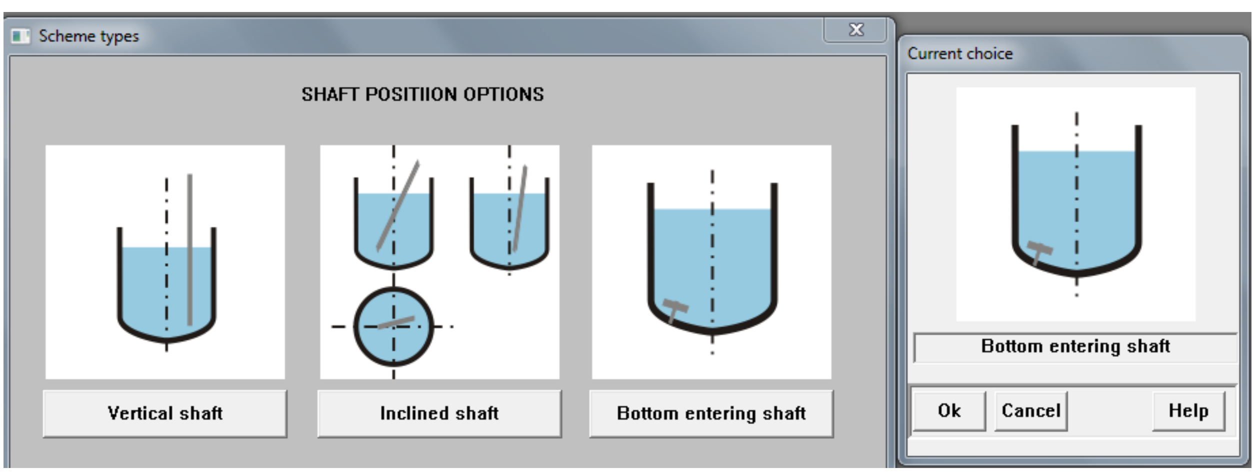


Figure 12. Shaft Position Options

There are three options for defining the shaft configuration (see Fig. 12).

Option 1. A vertical shaft

Select this option and click **OK**. The window for defining of the shaft position is displayed, see Fig.13.

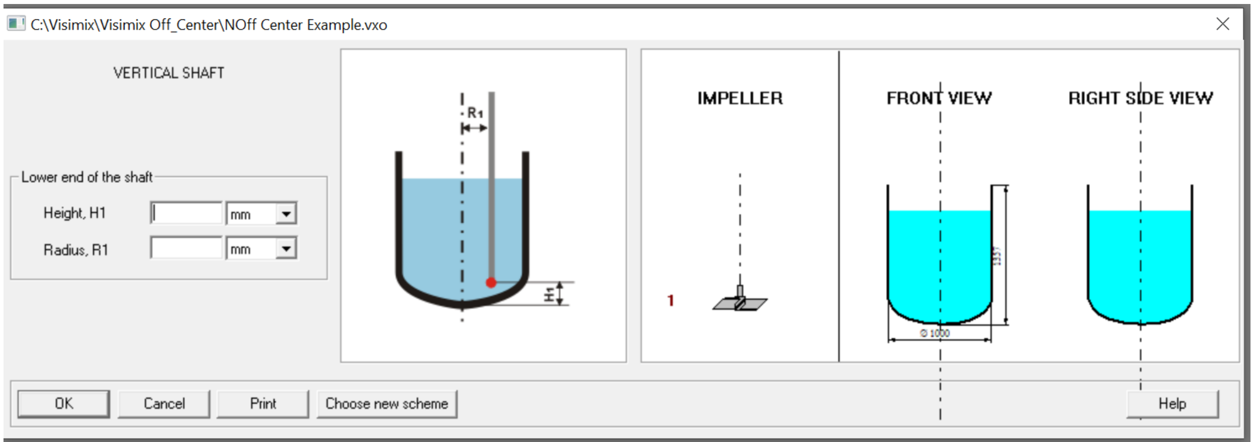


Figure 13. Shaft Position. Vertical Shaft Option before Input

Enter distance from the bottom to the shaft end, H1, and radial distance between the shaft and the tank axis, R1. Press **OK**. A sketch of the tank with the impeller is displayed, see Fig. 14.

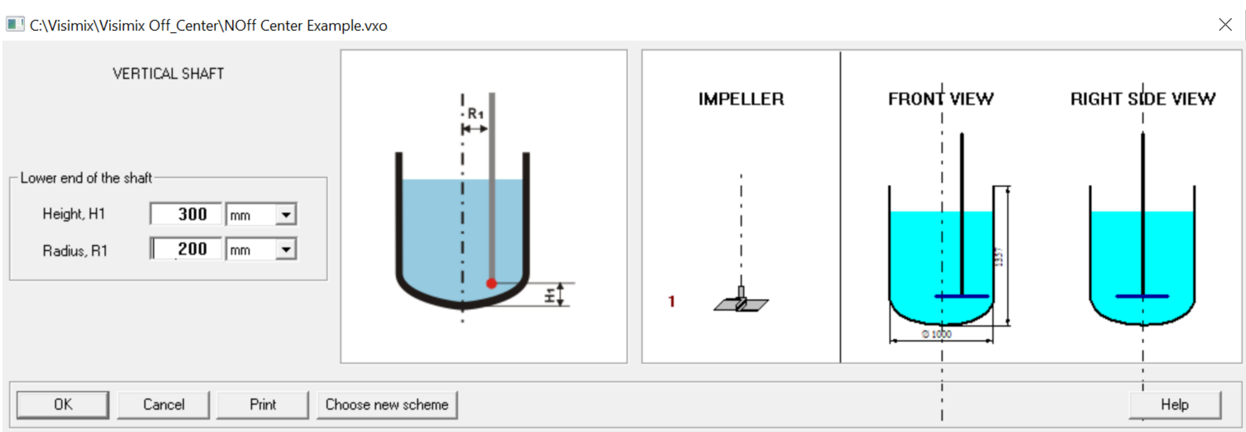


Figure 14. Shaft Position. Vertical Shaft Option after Input

Click **Chose new scheme** button. The Shaft Position Options window is displayed again as shown in Fig. 12.

Option 2. An inclined shaft

Select this option in Fig. 12 and click OK. The window for selecting shaft position is displayed as shown in Fig. 15.

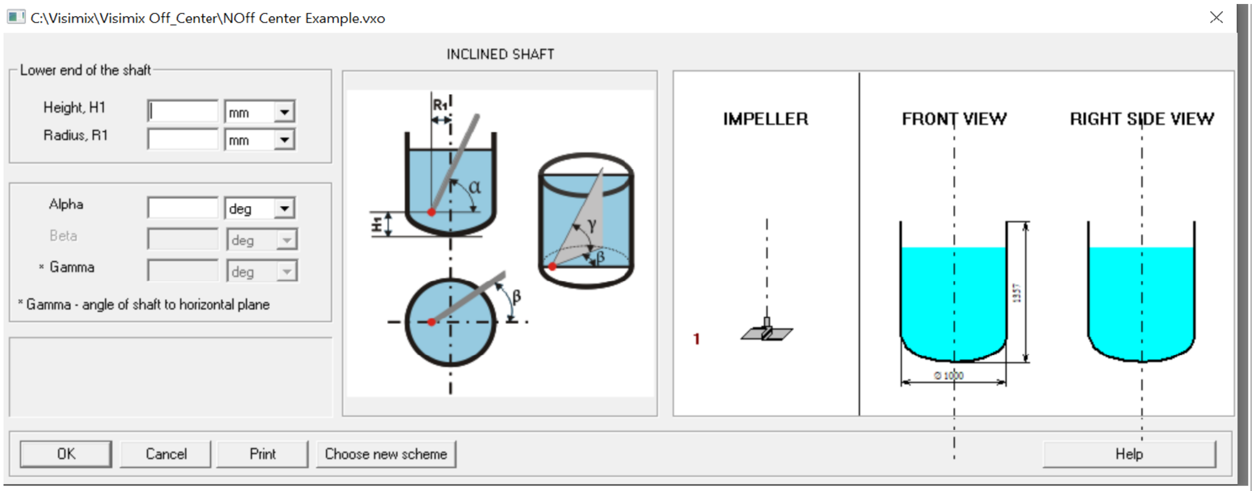


Figure 15. Shaft Position. Inclined Shaft Option before Input

Enter distance from the bottom to the shaft end, H1, radial distance between the shaft and tank axis, R1, and Alpha and Betta (or Gamma) shaft inclination angles as shown in the sketch in Fig. 15. A sketch of the tank with the impeller is displayed, see Fig. 16.

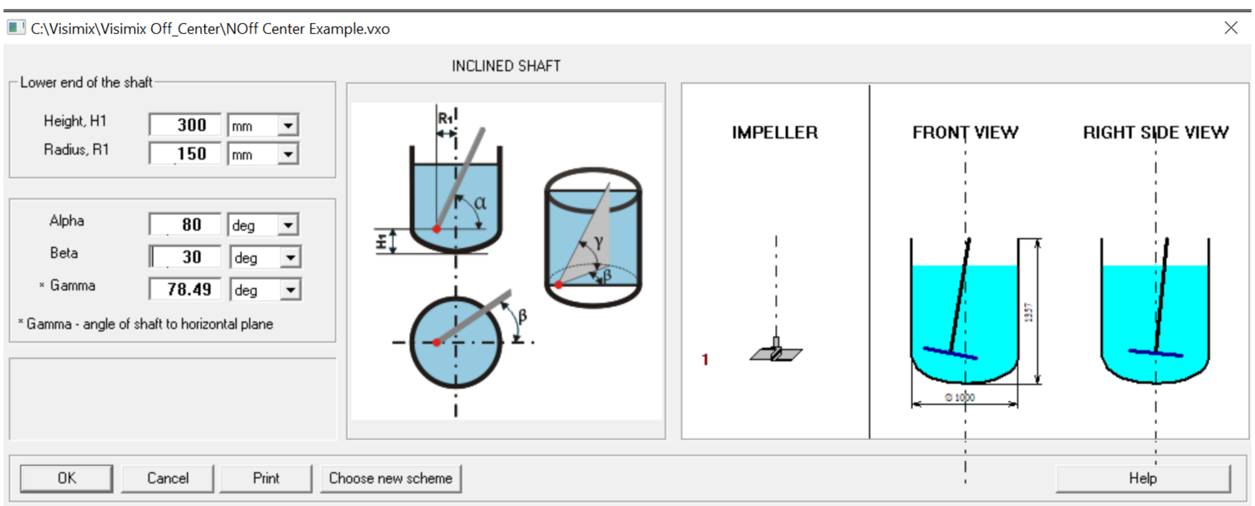


Figure 16. Shaft Position. Inclined Shaft Option after Input

Click the **Chose new scheme** button. The Shaft Position Options window is displayed again as shown in Fig. 12.

In this example, the third option, Bottom-entering shaft, is discussed.

Option 3. Bottom-entering shaft

NOTE:

This option is available for tanks with the elliptical bottom and the one-impeller configuration.

Select this option and click OK. The window for defining shaft position is displayed as shown in Fig.17.

Enter distance of the shaft entering point from the tank axis, R, and shaft length, L. A sketch of the tank with an impeller is displayed as shown in Fig. 17.

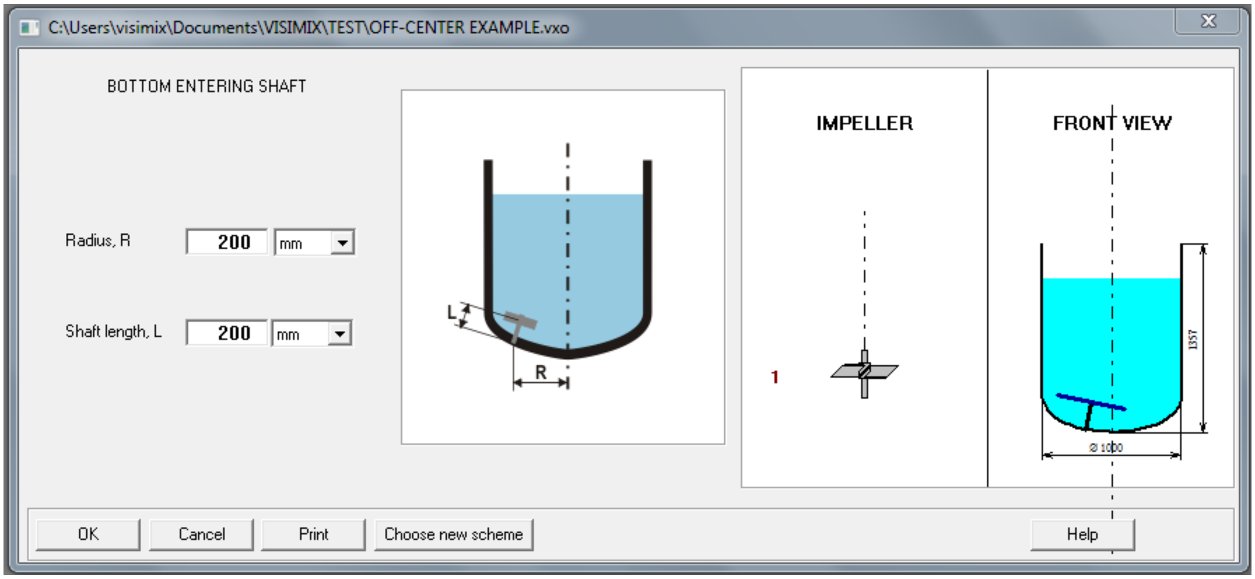


Figure 17. Shaft Position and Dimensions

Click **OK** and a sketch of the mixing tank is displayed as shown in Fig 18.

Figure 18. Mixing Tank Sketch

Step 7. Simulation of mixing parameters (optional)

Under **Calculate** tab of the main toolbar shown in Fig, 1, select **Hydrodynamics, Turbulence** or **Single-Phase Liquid Mixing** option from the drop-down menu shown in Fig. 19.

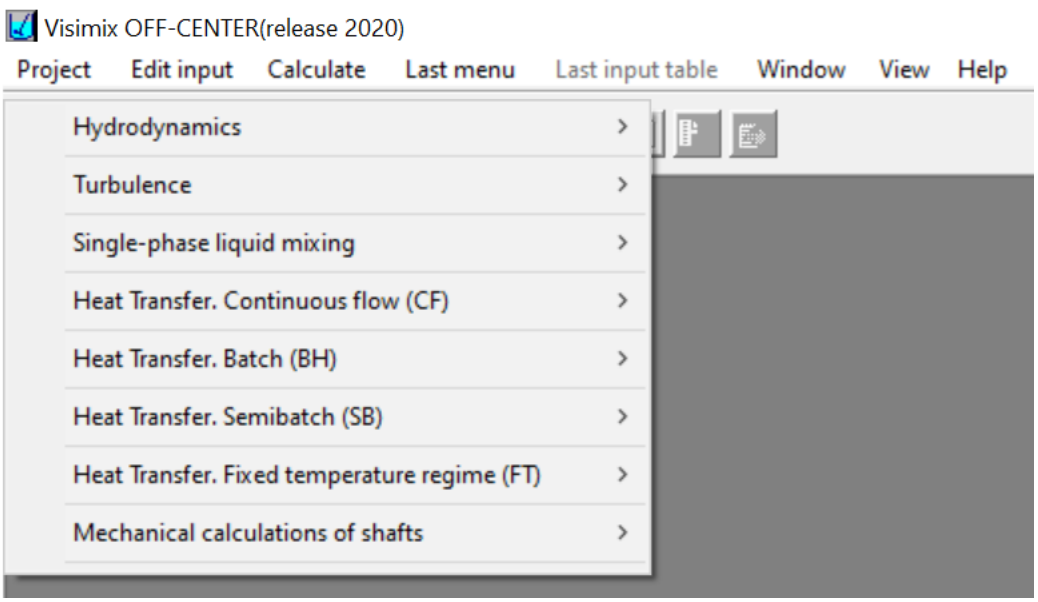


Figure 19. Drop-down Menu under Calculate Tab

Click **Hydrodynamics** (this is an option that we discuss below as an example) to open the drop-down of turbulence parameters shown in Fig. 20.

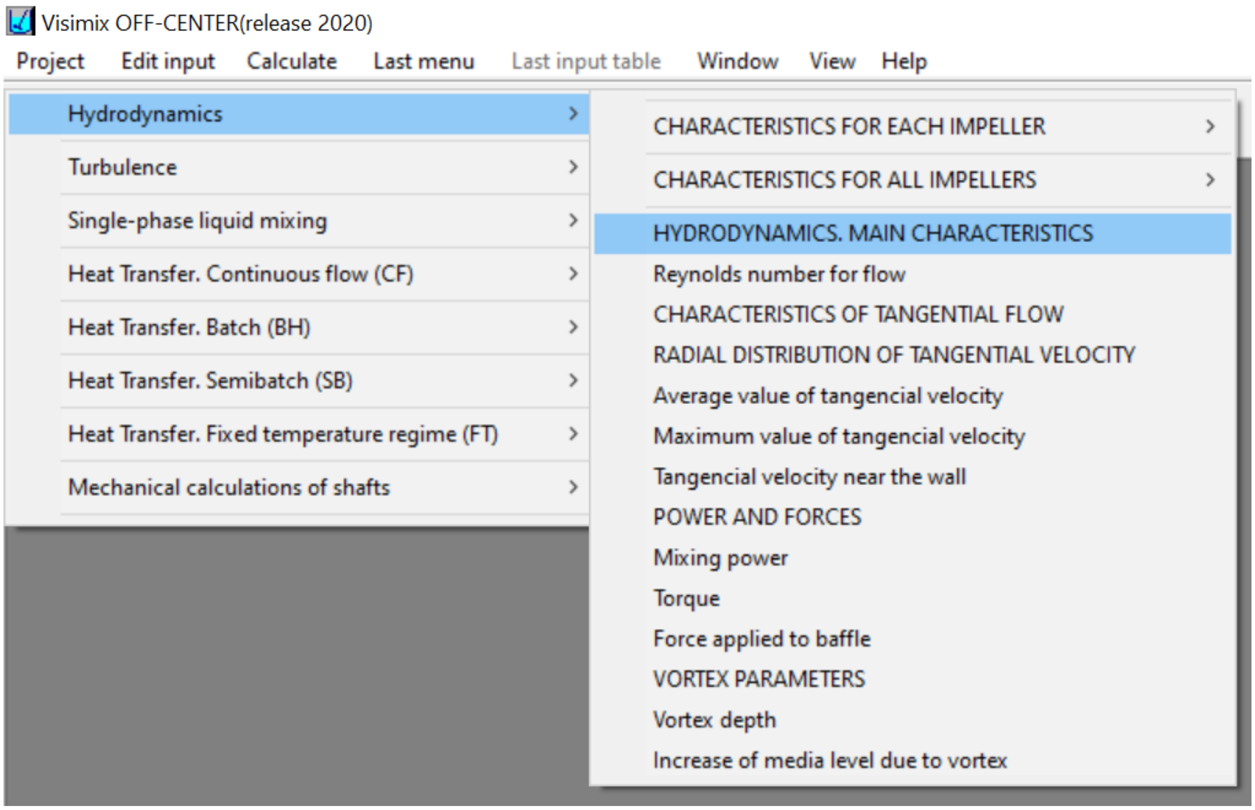
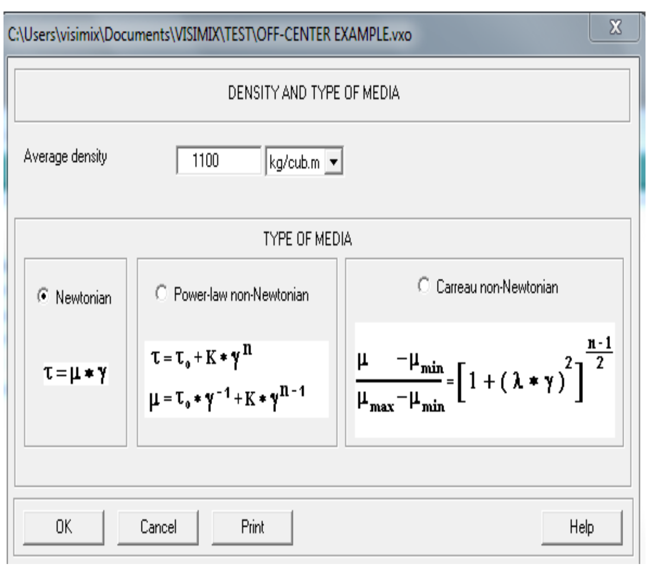


Figure 20. Drop-down Menu for **Hydrodynamics** Options

Click a turbulent characteristic of interest. The first input window shown in Fig. 21 is displayed. In this example, enter **Average density** of 1100 kg/m^3 and **Newtonian** for **Type of Media**. The second input window shown in Fig. 22 is displayed. In this example, enter **Viscosity** of 2 **cP**.



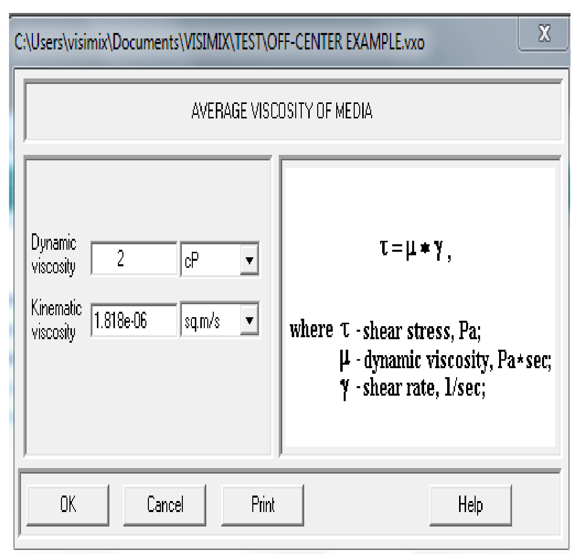
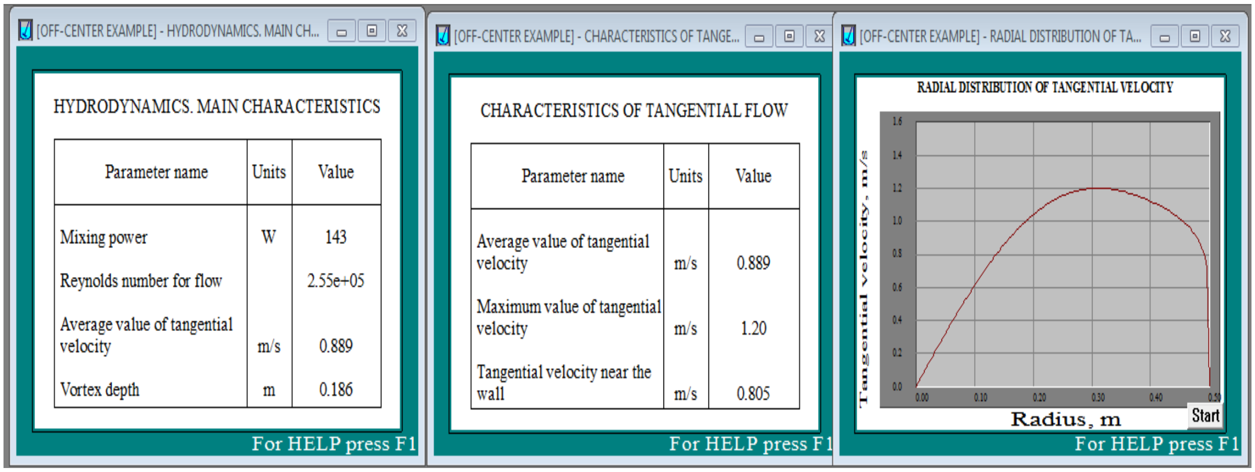


Figure 21. Density and Type of Media Figure 22. Average Viscosity of Media

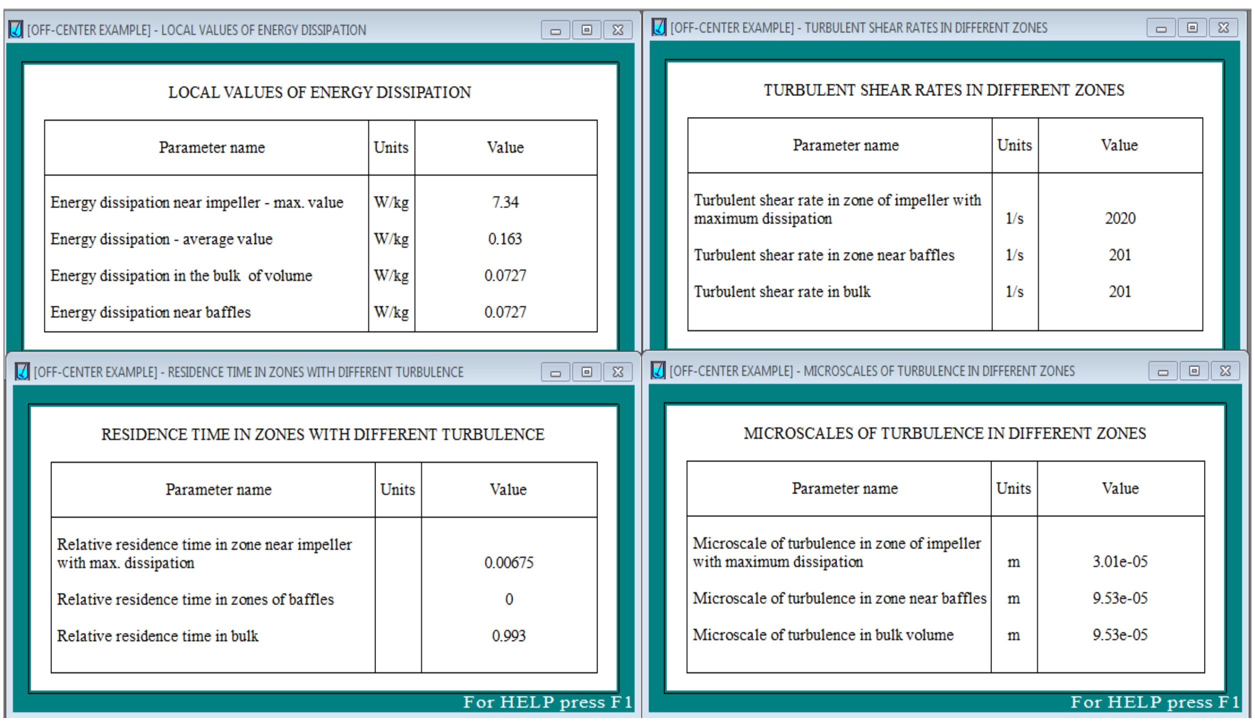
Press **OK** and the output characteristic requested is displayed instantly. Continue selecting output characteristics of interest one-by-one. They are displayed in the output windows as shown below in Figs. 23 through 25 in addition to the optional output characteristics previously requested.



Parameter name	Units	Value
Mixing power	W	143
Reynolds number for flow		2.55e+05
Average value of tangential velocity	m/s	0.889
Vortex depth	m	0.186

Parameter name	Units	Value
Average value of tangential velocity	m/s	0.889
Maximum value of tangential velocity	m/s	1.20
Tangential velocity near the wall	m/s	0.805

Figure 23. Hydrodynamics. Selected Output Characteristics



Parameter name	Units	Value
Energy dissipation near impeller - max. value	W/kg	7.34
Energy dissipation - average value	W/kg	0.163
Energy dissipation in the bulk of volume	W/kg	0.0727
Energy dissipation near baffles	W/kg	0.0727

Parameter name	Units	Value
Turbulent shear rate in zone of impeller with maximum dissipation	1/s	2020
Turbulent shear rate in zone near baffles	1/s	201
Turbulent shear rate in bulk	1/s	201

Parameter name	Units	Value
Relative residence time in zone near impeller with max. dissipation		0.00675
Relative residence time in zones of baffles		0
Relative residence time in bulk		0.993

Parameter name	Units	Value
Microscale of turbulence in zone of impeller with maximum dissipation	m	3.01e-05
Microscale of turbulence in zone near baffles	m	9.53e-05
Microscale of turbulence in bulk volume	m	9.53e-05

Figure 24. Turbulence. Selected Output Characteristics

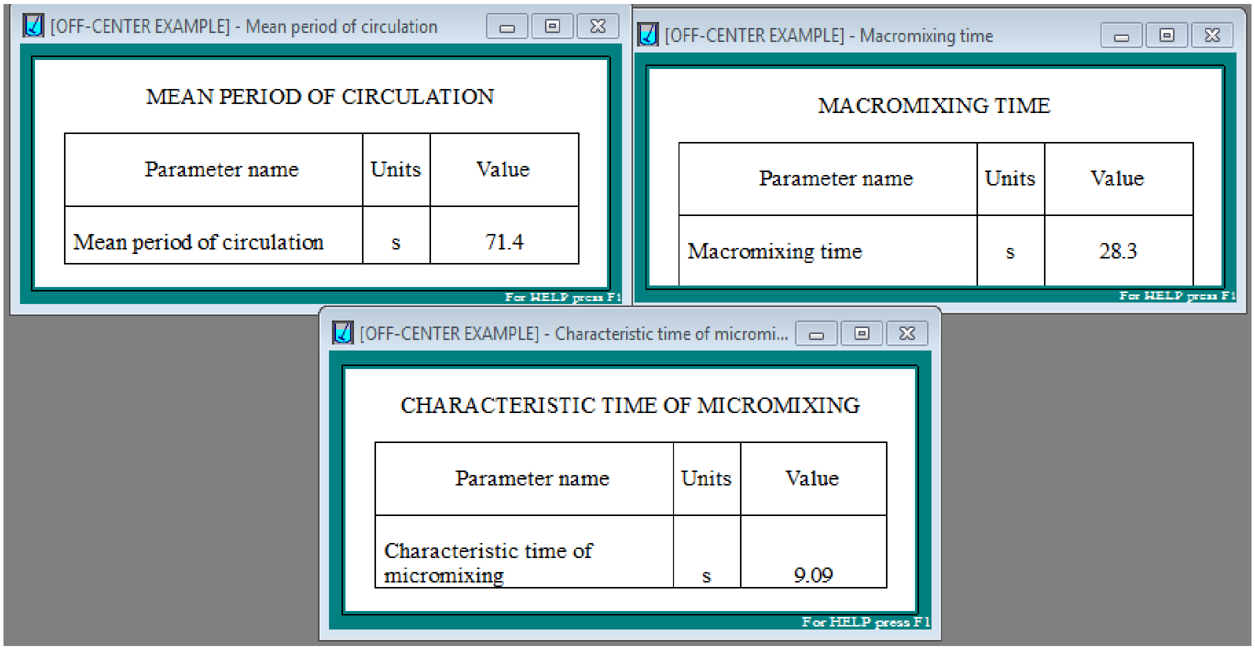


Figure 25. Single-Phase Liquid Mixing. Selected Output Characteristics

Step 9. Simulation of heat transfer parameters (optional)

VisiMix OFF-CENTER tool also supports simulation of heat transfer characteristics for tanks operated under different heating or cooling conditions.

It can utilize the entire scope of heat transfer simulations supported by VisiMix Turbulent for tanks with liquid- or gas-fed heating/cooling jackets of the following configurations:

- Simple or sectional jackets;
- Jackets with heat transfer enhancing hardware;
- Half-pipe coil jackets; or
- Embossed/ dimpled jackets.

The tool can be used for simulation of various cooling/heating modes of tank operation, such as:

- Steady state or batch dynamics;
- Semi-batch or continuous flow heat transfer operations;
- Temperature-dependent chemical reactions with heat release and/or consumption.

In addition, the tool provides access to the heat-transfer oriented VisiMix databases of tank materials, fouling characteristics, and physical properties of various liquids and heating/cooling fluids.

Step 10. Mechanical analysis of a shaft (optional)

Options for calculation of torsion shear and shaft vibration characteristics are available.