Lesson 2: Basic Functionality of COSMOSFloWorks

Goals of This Lesson

❑ Introduce flow analysis as a tool predicting characteristics of various flows over and inside 3D objects modeled by SolidWorks and thereby solving various hydraulic and gasdynamic engineering problems. Upon successful completion of this lesson, the students should understand COSMOSFloWorks' basic approaches to solving hydraulic and gasdynamic engineering problems. The students should see that, analyzing the flow over complex objects can influence the objects' design and performance, significantly save time and money by performing a properly-stated comprehensive CFD analysis with COSMOSFloWorks instead of conducting extremely time-consuming and expensive experimental works.

❑ Associate COSMOSFloWorks' flow analysis as "a chess game", in which it is very easy "to arrange the figures over the board before the game", some effort is required "to obey the game rules", and it will be necessary to apply some strategy "to win the game", i.e. to obtain correct and accurate results. The students should see that, due to COSMOSFloWorks' clear and well-structured interface, it is relatively easy "to arrange the figures over the board before the game", therefore the user will have more time to develop a strategy of solving the engineering problem, specify the boundary conditions properly, and study the obtained results for a possible change in strategy. So, this step shows how "to arrange the figures over the board before the game" in COSMOSFloWorks.

❑ Show the students the proper ways of correctly simulating real objects and flow phenomena with COSMOSFloWorks.
The results of analysis may slightly vary depending on versions/builds of SolidWorks and COSMOSFloWorks.

Outline

- ❑ In Class Discussion

- ❑ Active Learning Exercise — Determination of Hydraulic Loss
  - Opening the Valve.SLDPRT Document
  - Checking the COSMOSFloWorks Menu
  - Model Description
  - Creating Lids
  - Creating a Project
  - COSMOSFloWorks Design Tree
  - Specifying Boundary Conditions
  - Specifying Surface Goals
  - Specifying the Equation Goal
  - Running the Calculation
  - Monitoring the Solver
  - Accessing the Results
  - Creating a Cut Plot
  - Displaying Flow Trajectories
  - Creating a Goal Plot
  - Cloning Project
  - Changing the Valve Angle
  - Changing the Geometry Resolution
  - Changing the Computational Domain
  - Getting the Valve's Hydraulic Loss

- ❑ 5 Minute Assessment – Answer Key

- ❑ In Class Discussion — Changing the Inlet Boundary Condition

- ❑ More to Explore — Modifying the Geometry

- ❑ Exercises and Projects — Hydraulic Loss Due to Sudden Expansion

- ❑ Lesson Summary

In Class Discussion

Ask students where a fluid flow and heat transfer analysis software can be beneficial for a design engineer?

Answer

❑ Machinery: Hydraulic/pneumatic systems manufacturers can improve their designs regarding flow distribution and pressure drop. Oil industry can better understand flow through valves or mixing vessels, etc. Particle tracking can be used to understand how safe equipment is against erosion.

❑ Electrical and Electronics: Designers of electronic devices (computers, audio/video, etc.) can check for efficient cooling by simulating convection and conduction within their designs.

❑ Aerospace and Automotive: Land-, air- and marine-vehicle designers can achieve maximum performance, at least cost: Manifolds, brake systems, engine cooling jacket, flow around a wing or through a rocket nozzle, flow around an immersed body etc.

❑ HVAC & Building: HVAC equipment manufactures can optimize product performance: flow through ducts, heat exchangers, flow and temperature distributors in rooms to determine duct locations, etc.

❑ Consumer Products: Consumers designers can improve the uniform distribution in an oven or correct the flow distribution in a dishwasher etc.

❑ Design engineers, analysts, and other professionals can recognize forces and torques and other loads acting on objects due to the fluid flow and use this knowledge in further structural analysis for achieving the better designs.

More to explore

Regarding structural analysis, ask the students how the forces acting on a particular object (whose stress is analyzed within COSMOSWorks) was determined. Are these forces always known or estimated from known formulas?

Answer

❑ In some problems, even involving fluids, these forces are either well known or can be neglected. For example, a force acting on the chair's legs is determined as the weight of the student sitting on it in a room plus the chair's weight, or a force and moment acting on a manually operated small valve can be neglected. But determining the forces on many problems in industry are just too complicated and computer computations will be required to determine the needed forces. For example, if the valve is large, e.g. as used for hydroelectric power stations, both the force and the moment acting on the valve from the fluid must be certainly taken into account, otherwise the valve's parts (e.g. bearings) and devices (e.g. actuators turning the valve) can fail, so the valve becomes inoperative.

Active Learning Exercise — Determination of Hydraulic Loss

Use COSMOSFloWorks to perform fluid internal analysis on the `Valve.SLDPRT` part shown to the right.

The step-by-step instructions are given below.
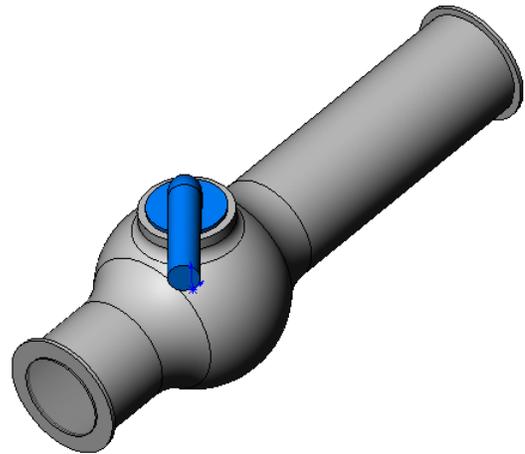
Opening the Valve.SLDPRT Document

We recommend that you save the **Education Examples** to a temporary directory to save the original copy for repeated use.

1 Create a temporary directory named **temp** in the **Examples** folder of the COSMOSFloWorks installation directory.

2 Copy the **Education Examples** directory into the **temp** directory.

3 Click **File**, **Open**. In the **Open** dialog box, browse to the `Valve.SLDPRT` part located in the **temp\Education Examples\Hydraulic Loss** folder and click **Open** (or double-click the part).

Checking the COSMOSFloWorks Menu

If COSMOSFloWorks is properly installed, the `FloWorks` menu appears on the SolidWorks' menu bar. If not:

File   Edit   View   Insert   Tools   FloWorks   Window   Help

COSMOSFloWorks menu

1 Click **Tools**, **Add-Ins**.

The **Add-Ins** dialog box appears.

2 Check **COSMOSFloWorks**. If COSMOSFloWorks is not in the list, you need to install COSMOSFloWorks.

3 Click **OK**. The `FloWorks` menu appears on the SolidWorks' menu bar.

Model Description

This is a ball valve. Turning the handle closes or opens the valve.

The local hydraulic loss (or resistance) produced by a ball valve installed in a piping system depends on the valve design dimensions and on the handle turning angle. The ball-to-pipe diameter ratio governs the handle turning angle at which the valve becomes closed.
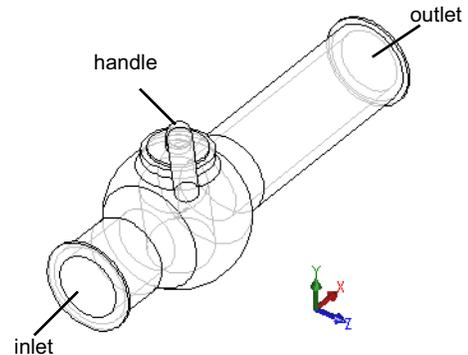
The standard engineering definition of a hydraulic resistance of an obstacle in a pipe is the difference between the total pressures (i.e. where a stream is not disturbed by the obstacle) upstream and downstream of the obstacle (the valve in our case) divided by the incoming dynamic head, from which the hydraulic resistance due to the friction over the pipe section is subtracted.

In this example we will obtain the local hydraulic resistance of the ball valve whose handle is turned by an angle of 40°. The Valve analysis represents a typical COSMOSFloWorks internal analysis.

Note: Internal flow analyses are analyses where fluid enters a model at the inlets and exits the model through outlets with the exception of some natural convection problems that may not have openings.

To perform an internal analysis all the model openings must be closed with lids, which are needed to specify inlet and outlet flow boundary conditions on them. In any case, the internal model space filled with a fluid must be fully closed. You simply create lids as additional extrusions covering the openings.

Creating Lids

Creating Inlet Lid

1 Select the face shown in the picture.

2 Click **Sketch** on the Sketch toolbar.

3 Select the tube's edge.

4 Click **Convert Entities** on the Sketch tools toolbar.

5 Complete the sketch by clicking **OK** button in the confirmation corner of the graphics area.
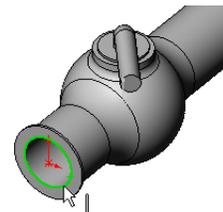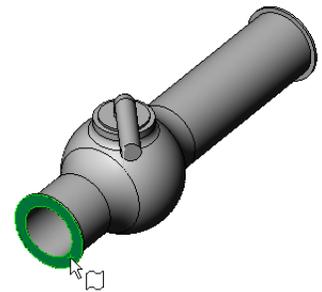
6 Click **Extrude Boss/Base** on the Features toolbar.

7 In the **Extrude Feature** PropertyManager change the settings as shown.

   • End Condition = **Mid Plane**

   • Depth = 0.005

8 Click **OK** to create the inlet lid.
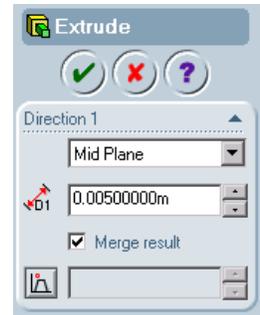
Next, in the same manner we will create the outlet lid.

Creating Outlet Lid

9 Select the face shown in the picture.

10 Click **Sketch** on the Sketch toolbar.
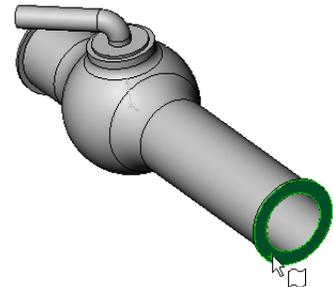
11 Select the tube's edge.

12 Repeat the steps 3 to 8 to create the lid at outlet.

13 Rename the new extrusions Extrude1 and Extrude2 to **Inlet Lid** and **Outlet Lid** correspondingly.
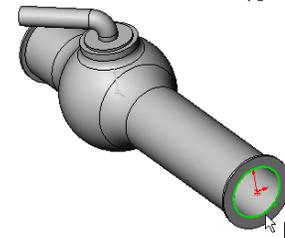
Not sure you have created the lids properly? COSMOSFloWorks can easily check your model for possible geometry problems.

Checking the Geometry

1 To ensure the model is fully closed, click **FloWorks**, **Tools**, **Check Geometry**.

2 Click **Check** to calculate the fluid volume of the model. If the fluid volume is equal to zero, the model is not closed.

> **Note:** This **Check Geometry** tool allows you to calculate the total fluid and solid volumes, check bodies for possible geometry problems (i.e. tangent contact) and visualize the fluid area and solid body as separate models.
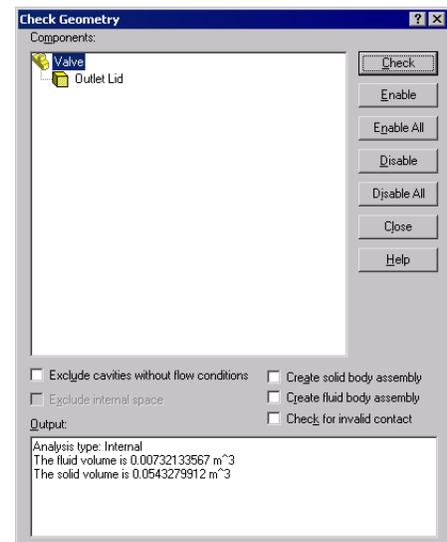
The first step in performing flow analysis is to create a COSMOSFloWorks project.

Creating a Project

1 Click **FloWorks**, **Project**, **Wizard**. The project wizard guides you through the definition of a new COSMOSFloWorks project.

2 In the **Project Name** dialog box, click **Use current** (40 degrees).

   Each COSMOSFloWorks project is associated with a SolidWorks configuration. You can attach the project either to the current SolidWorks configuration or create a new SolidWorks configuration based on the current one.

   Click **Next**.

3 In the **Units** dialog box you can select the desired system of units for both input and output (results).

   For this project use the International System **SI** by default.

   Click **Next**.

4 In the **Fluid Type and Physical Features** dialog box you can select the fluid type.

   The selected fluid type is assigned for all fluids in the analysis. This dialog also allows you to specify advanced physical features you want to take into account: heat transfer in solids, high Mach number gas flow, gravitational effects, time settings, laminar only flow, surface-to-surface radiation.

   Specify **Liquid** and do not use any of the above-mentioned features.

   Click **Next**.

5 In the **Analysis Type** dialog box you can select either **Internal** or **External** type of the flow analysis.

   Specify **Internal** type and accept the other default settings.

   Click **Next**.
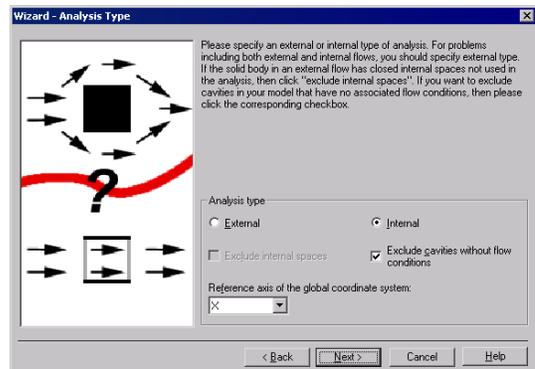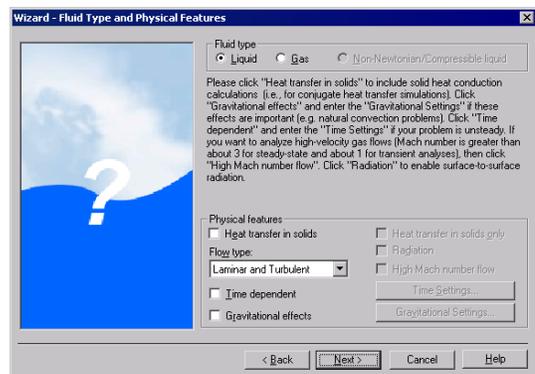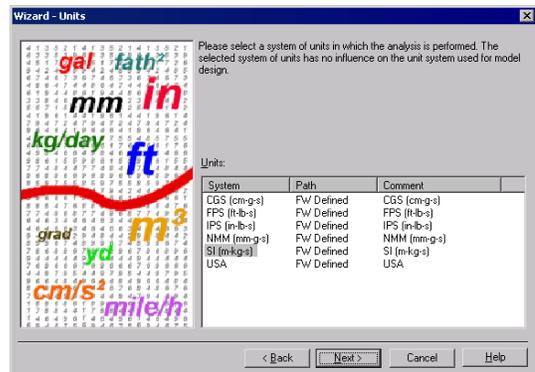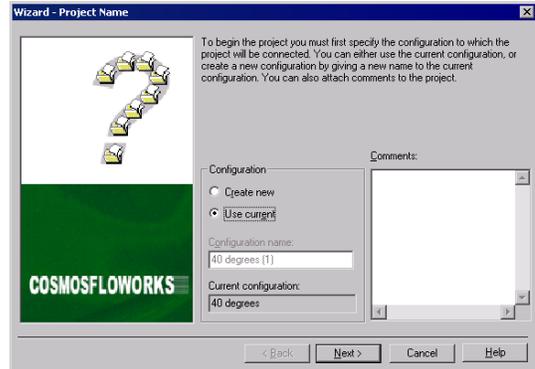
6 In the **Roughness** dialog box you can specify the wall roughness value applied by default to all model walls. To specify the roughness value for a specific wall you must create a **Real Wall** boundary condition.

In this project we will not concern rough walls.

Click **Next**.
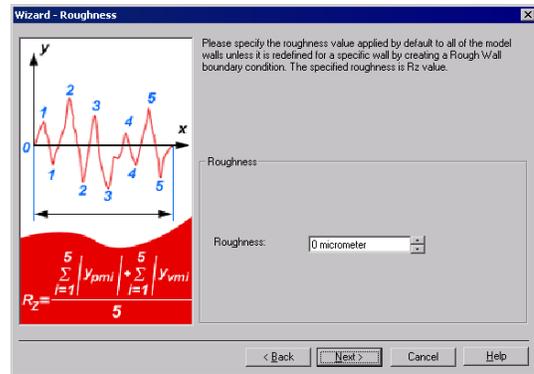
7 In the **Selecting Fluid Substances** dialog box you can select the fluid substances whose flow is analyzed in the COSMOSFloWorks project.

Since we use water in this project, double-click the **Water SP** item in the **Database of fluids** list taken from COSMOSFloWorks' **Engineering Database**.

Click **Next**.
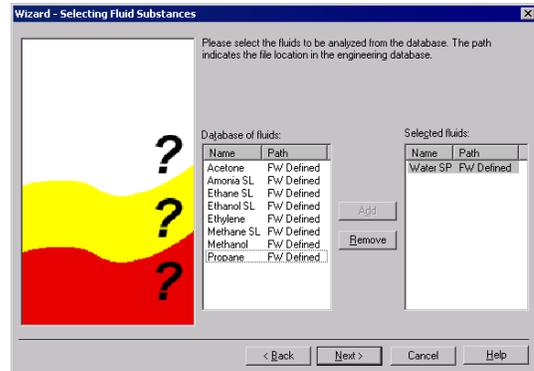
Note: Engineering Database contains numerical physical information on a wide variety of gas, liquid and solid substances as well as radiative surfaces. You can also use the Engineering Database to specify a porous medium. The Engineering Database contains pre-defined unit systems. It also contains fan curves defining volume or mass flow rate versus static pressure difference for selected industrial fans. You can easily create your own substances, units, fan curves or specify a custom parameter you want to visualize.

SP (Standard Pressure) means that temperature dependencies of the liquid are taken along the standard isobar P=0.1 MPa.

---

8 Since we do not intend to calculate heat transfer in solids, the **Default Wall Conditions** dialog box appears, where you must specify the thermal wall boundary conditions applied by default to all the model walls contacting with the fluid.

For this project accept the default **Adiabatic wall** feature denoting that all the model walls are heat-insulated.

Click **Next**.

9 In the **Initial and Ambient Conditions** dialog box specify initial values of the flow parameters. For steady internal problems these values specified closer to the expected flow field will reduce the analysis time.

For this project use the default values.

Click **Next**.
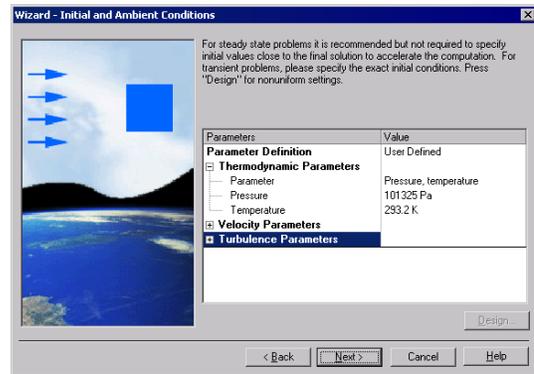
Note: For steady flow problems COSMOSFloWorks iterates until the solution convergences. For unsteady (transient, or time-dependent) problems COSMOSFloWorks marches in time for a period you specify.

---

10 In the **Result Resolution and Geometry Resolution** dialog box you can control the analysis accuracy as well as the mesh settings and, through them, the required computer resources (CPU time and memory).

For this project accept the default result resolution level 3.
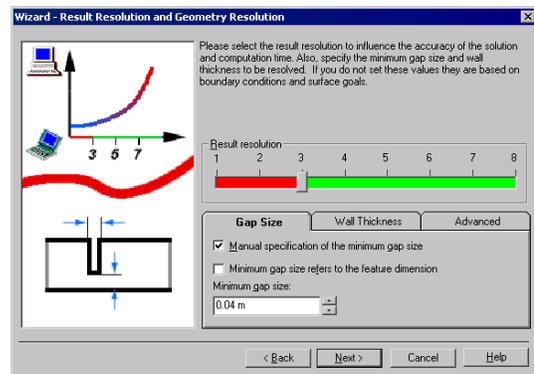
Result Resolution governs the solution accuracy that can be interpreted as resolution of calculation results. You specify result resolution in accordance with the desired solution accuracy, available CPU time and computer memory. Because this setting has an influence on the number of generated mesh cells, a more accurate solution requires longer CPU time and more computer memory.

Geometry Resolution (specified through the minimum gap size and the minimum wall thickness) governs proper resolution of geometrical model features by the computational mesh. Naturally, finer Geometry Resolution requires more computer resources.

Select the **Manual specification of the minimum gap size** check box and enter `0.04 m` for the minimum flow passage.
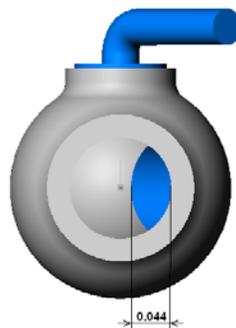
Note: COSMOSFloWorks calculates the default minimum gap size and minimum wall thickness using information about the overall model dimensions, the computational domain, and faces on which you specify conditions and goals. However, this information may be insufficient to recognize relatively small gaps and thin model walls. This may cause inaccurate results. In these cases, the Minimum gap size and Minimum wall thickness must be specified manually.

Click **Next**.

11 The **Summary** dialog box presents you brief general information about the project specified in the **Wizard**. Click **Finish**.

COSMOSFloWorks Design Tree

After the basic part of the project has been created, a new COSMOSFloWorks design tree tab ▣ appears on the right side of the Configuration Manager tab.

Note: The COSMOSFloWorks Design Tree provides a convenient specification of project data and view of results. You also can use the COSMOSFloWorks design tree to modify or delete the various COSMOSFloWorks features.

At the same time, in the SolidWorks graphics area a computational domain wireframe box appears. The flow and heat transfer calculations are performed inside the computational domain. The computational domain is a rectangular prism for both the 3D analysis and 2D analysis. The Computational Domain boundaries are parallel to the Global Coordinate System planes.

computational domain
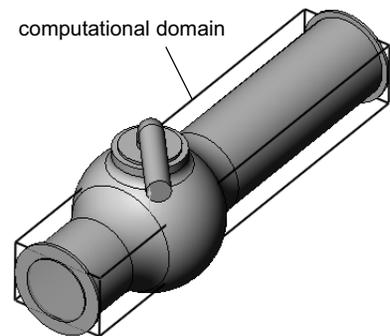
Now let us specify the other parts of the project.

The next step is specifying Boundary Conditions. Boundary Conditions are used to specify fluid characteristics at the model inlets and outlets in an internal flow analysis or on model surfaces in an external flow analysis.

Specifying Boundary Conditions

1 Click **FloWorks**, **Insert**, **Boundary Condition**.

2 Select the **Inlet Lid** inner face (in contact with the fluid). To access the inner face, right-click the lid's outer face and choose **Select Other**. Right-click the mouse to cycle through the faces under the cursor until the inner face is highlighted, then click the left mouse button.
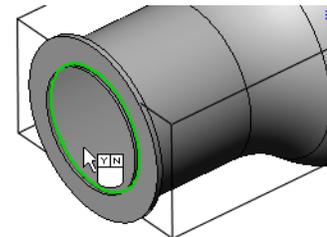
The selected face appears in the **Faces to apply boundary condition** list.

3 In the **Type of boundary condition** list, select the **Inlet Velocity** item, and then click the **Settings** tab.
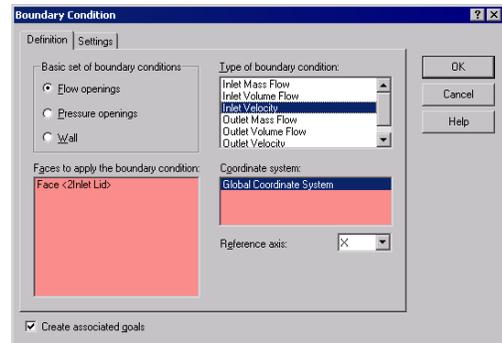
4 Double-click the **Value** cell of the **Velocity normal to face** item and set it equal to 1 (type the value, the units appears automatically).

Accept all the other parameters and click **OK**.
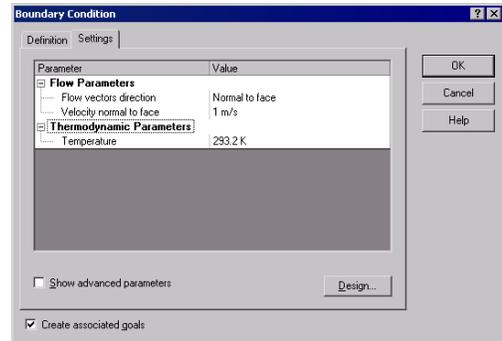
By specifying this condition we defined at the ball valve pipe inlet the water enters the valve with the velocity of 1.0 m/s.

5 Select the **Outlet Lid** inner face.

In the graphics area, right-click outside the model and select **Insert Boundary Condition**. The **Boundary Condition** dialog box appears with the selected face in the **Faces to apply boundary condition** list.
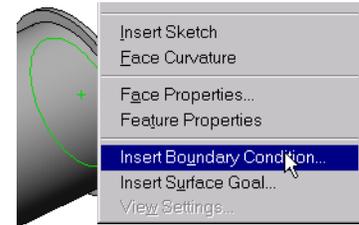
Let us specify pressure on this boundary, otherwise the problem specification is deficient. Before the calculation starts, COSMOSFloWorks checks the specified boundary conditions for mass flow rate balance. The specification of boundary conditions is incorrect if the total mass flow rate on the inlets is not equal to the total mass flow rate on the outlets. In such case the calculation will not start. Also, note that the mass flow rate value is recalculated from the velocity or volume flow rate value specified on an opening. Specifying at least one Pressure opening condition allow us to avoid problems with mass flow rate balance, since the mass flow rate on a Pressure opening is not specified but calculated during solving the problem.

6 Click **Pressure openings** and in the **Type of boundary condition** list, select the **Static Pressure** item. Click the **Settings** tab.
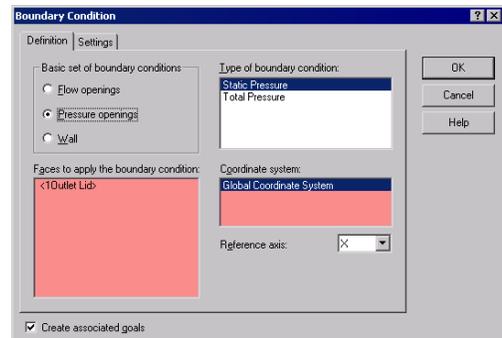
7 Accept the default values for static pressure (101325 Pa), temperature (293.2 K) and all the other parameters.
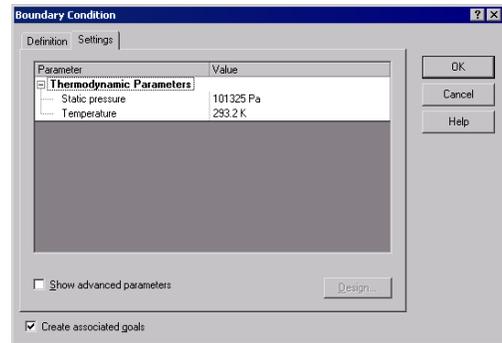
8 Click **OK**.

By specifying this condition we defined at the ball valve pipe exit that the water has a static pressure of 1 atm.

The model's hydraulic losses x is calculated as the difference between the model's inlet total pressure and the outlet total pressure, DP, divided by the dynamic pressure (dynamic head) determined at the model inlet:

$$x = (dP) / \frac{rV^2}{2} = (dP) / P_{dyn}$$

where r is water density, and V is inlet water velocity, P_{dyn} is the dynamic pressure at inlet. Since we already know (specify) the water velocity $(1 \frac{m}{s})$ and the water density (998.1934 for the specified temperature of 293.2), then our goal is to determine the total pressure value at the valve's inlet and outlet.

The easiest and fastest way to find the interested parameter is to specify the corresponding engineering goal.

Engineering goals are the parameters in which the user is interested. Setting goals is in essence a way of conveying to COSMOSFloWorks what you are trying to get out of the analysis, as well as a means of reducing the time COSMOSFloWorks takes to reach a solution. By only selecting the variable which the user desires accurate values for, COSMOSFloWorks knows which variables are important to converge upon (the variables selected as goals) and which can be less accurate (the variables not selected as goals) in the interest of time. Goals can be set throughout the entire domain (Global Goals), in a selected area (Surface Goal) or within a selected volume (Volume Goal). Furthermore, COSMOSFloWorks can consider the average value, the minimum value or the maximum value for goal settings. You can also define an Equation Goal that is a goal defined by an equation (basic mathematical functions) with the existing goals as variables. The equation goal allows you to calculate the parameter of interest (i.e., pressure drop) and keeps this information in the project for later reference.

Specifying Surface Goals

1 In the COSMOSFloWorks design tree, right-click the **Goals** icon and select **Insert Surface Goal.**

2 Select the inner face of the **Inlet Lid**.

To easily select a face, simply click the **Inlet Velocity1** item in the COSMOSFloWorks design tree. The face belonging to the specified boundary condition is automatically selected and appears in the **Faces to apply the surface goal** list.

3 In the **Goal type** list, select **Total Pressure**.

4 Accept the **Average Value** and agree to **Use the goal for convergence control**.

5 Click **OK**.

6 In the COSMOSFloWorks design tree click-pause-click the new **SG Average Total Pressure1** item and rename it to SG Average Total Pressure Inlet.
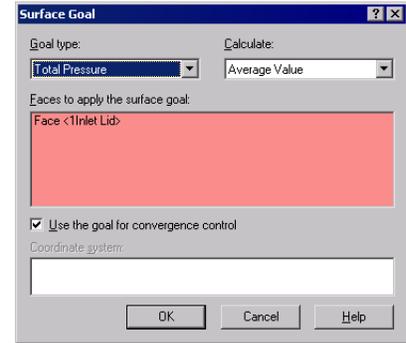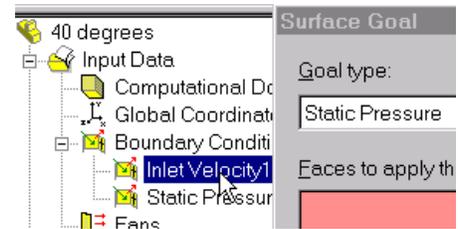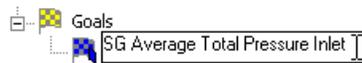
> **Note:** Another way to rename an item is to right-click the item and select **Properties**.

7 Right-click the **Goals** icon again and select **Insert Surface Goal**.

8 Click the **Static Pressure1** item to select the inner face of the **Outlet Lid.**

9 In the **Goal type** list, select **Total Pressure**.

10 Accept the **Average Value** and click **OK**.

11 Click-pause-click the new **SG Average Total Pressure1** item and rename it to SG Average Total Pressure Outlet.

12 Right-click the **Goals** icon again and select **Insert Surface Goal**.

13 Click the **Inlet Velocity1** item to select the inner face of the **Inlet Lid.**

14 In the **Goal type** list, select **Dynamic Pressure**.

15 Accept the **Average Value** and click **OK**.

16 Click-pause-click the new **SG Average Dynamic Pressure1** item and rename it to SG Average Dynamic Pressure Inlet.

The value of the dynamic pressure at the inlet can be calculated manually. We specified the dynamic pressure goal just for convenience in the further calculation for hydraulic losses.

After finishing the calculation you will need to manually calculate the hydraulic loss x from the obtained total pressures values. Instead, let COSMOSFloWorks make all the necessary calculations for you by specifying an Equation Goal.

Specifying the Equation Goal

Equation Goal is a goal defined by an analytical function of the existing goals. This goal can be viewed as equation goal during the calculation and while displaying results in the same way as the other goals. As variables, you can use any of the specified goals, including another equation goals, except for goals that are dependent on other equation goals. You can also use constants in the definition of the equation goal.

1 Right-click the **Goals** icon and select **Insert Equation Goal**. The **Equation Goal** dialog box appears.

2 Click the left bracket button (or type "(".

3 In the **Goal list** select the **SG Average Total Pressure Inlet** goal and click **Add**.

4 Click the minus button - or type "-".

5 In the **Goal list** select the **SG Average Total Pressure Outlet** goal and click **Add**.

6 Click the right bracket) and the forward slash / buttons, or type ")/".

7 In the **Goal list** double-click the **SG Average Dynamic Pressure Inlet** goal name.

8 In the **Dimensionality** list select **No units**.
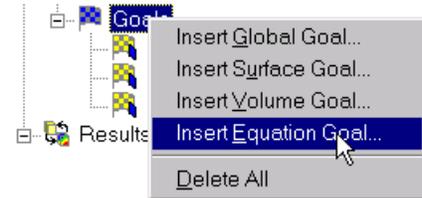
Note: To set an Equation Goal you can use only specified goals (including previously specified Equation Goals) and constants. If constants signify some physical parameters (i.e. length, area etc.) make sure of using the project's system of units. COSMOSFloWorks has no information about the physical meaning of the specified constant so you need to specify the displayed dimensionality yourself.

---

9 Click **OK**. The **Equation Goal 1** item appears in the tree.

10 Rename it to `Hydraulic Loss`.
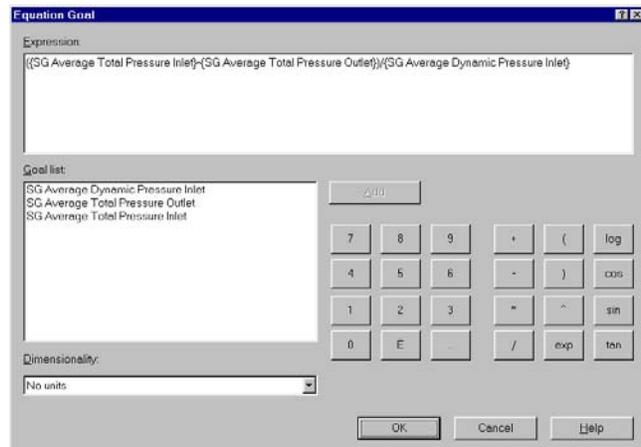
Now the COSMOSFloWorks project is ready for the calculation. COSMOSFloWorks will finish the calculation when the steady-state average value of total pressure calculated at the valve inlet and outlet are reached.

Running the Calculation

1 Click **FloWorks**, **Solve**, **Run**. The **Run** dialog box appears.

2 Click **Run** to start the calculation.
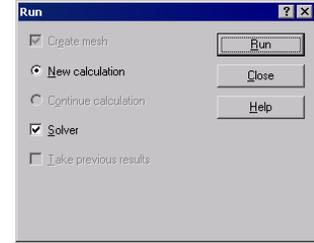
The calculation should take about 10 minutes to run on a 1.7 GHz PIV computer.

COSMOSFloWorks automatically generates a computational mesh in accordance with your settings of Result resolution and Geometry resolution. The mesh is created by dividing the computational domain into element rectangular volumes - cells. The cells are further subdivided as necessary to properly resolve the model geometry and flow features. This process is called mesh refinement. During the mesh generation procedure, you can see the current step and the mesh information in the **Mesh Generation** dialog box.
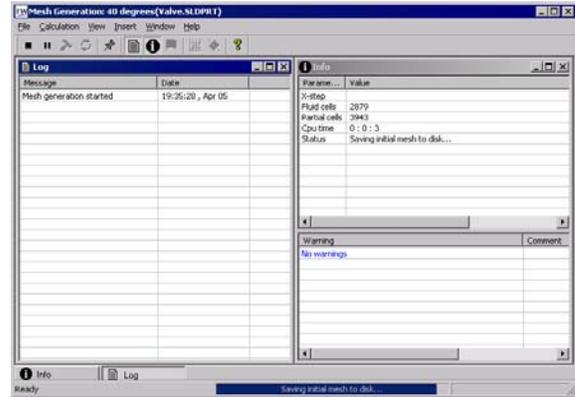
Monitoring the Solver

This is the solution monitor dialog box. On the left is a log of each step taken in the solution process. On the right is an information dialog box with mesh information and any warnings concerning the analysis.
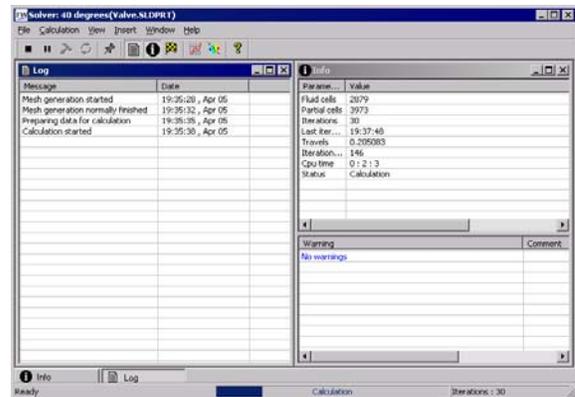
During the calculation you can monitor the convergence behavior of your goals (Goal Plot), view the current results in the specified plane (Preview) and displays the minimum and maximum parameter values at the current iteration (Min/Max table).

Creating Goal Plot

1 Click **Insert Goal Plot** on the Solver toolbar. The **Add/Remove Goals** dialog box appears.

2 Click **Add All** to add all goals to the **Selected goals** list and click **OK**.
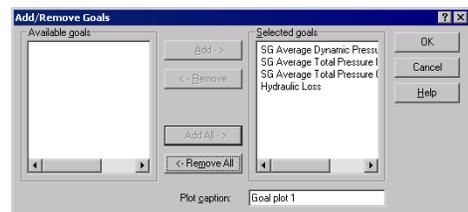
This is the Goals dialog box and each goal created earlier is listed above. Here you can see the current value and graph for each goal as well as the current progress towards completion given as a percentage. The progress value is only estimate and the rate of progress generally increases with time.
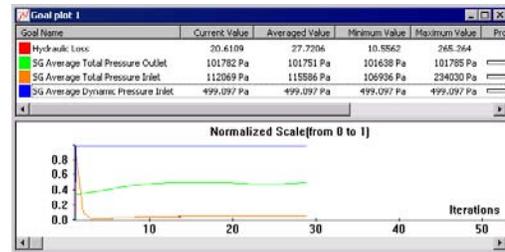
Preview Results

1 Click **Insert Preview** on the Solver toolbar. The **Preview Settings** dialog box appears.

2 Click the FeatureManager tab .
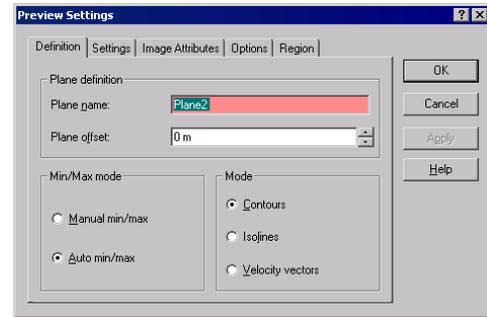
3 Select `Plane 2`.

   For this model `Plane 2` is a good choice to use as the preview plane. The preview plane can be chosen anytime from the Feature Manager.

4 Click **OK** to display the preview plot of the static Pressure distribution.
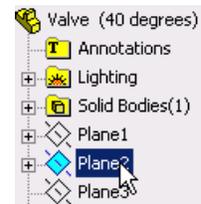
> **Note:** You can specify a parameter you want to display in the preview plane, the parameter range and display options for velocity vectors at the **Setting** tab of the **Preview Settings** dialog box.

The preview allows one to look at the results while the calculation is still running. This helps to determine if all the boundary conditions are correctly defined and gives the user an idea of how the solution will look even at this early stage.
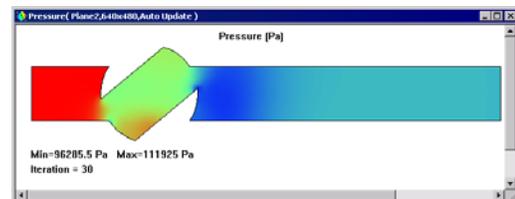
At the start of the run the results might look odd or change abruptly. However, as the run progresses these changes will lessen and the results will settle in on a converged solution. The result can be displayed either in contour-, isoline- or vector-representation.

> **Note:** Why does the static pressure increase at the local region inside the valve? This is due to a deceleration (up to stagnation within a small region) of the stream impacting the valve's wall in this region, so the stream's dynamic pressure is partly transformed into the static pressure while the stream's total pressure is nearly constant in this region, so the static pressure rises.

5 When the solver is finished, close the monitor by clicking **File**, **Close**.

Accessing the Results

1 Right-click the **Results** icon and select **Load Results** to activate the postprocessor. Select `1.fld` and click **Open**.

2 Click the plus sign (+) besides **Results**.

Once the calculation finishes, you can view the saved calculation results through numerous COSMOSFloWorks options in a customized manner directly within the graphics area. The Result options are: Cut Plots (section view of parameter distribution), Surface Plots (parameter distribution on a selected surface), Isosurfaces, Flow Trajectories (streamlines and particle trajectories), Goal Plot (behavior of the specified goals during the calculation), XY Plots (parameter change along a curve, sketch), Surface Parameters (getting parameters at specified surfaces), Point Parameters (getting parameters at specified points), Report (project report output into MS Word) and Animation of results.

Creating a Cut Plot

1 Right-click the **Cut Plots** icon and select **Insert**. The **Cut Plot** dialog box appears.

The Cut Plot displays results of a selected parameter in a selected view section. To define the view section, you can use SolidWorks planes or model planar faces (with the additional shift if necessary). The parameter values can be represented as a contour plot, as isolines, as vectors, or in a combination (e.g. contours with overlaid vectors).

2 Click the SolidWorks FeatureManager and select `Plane2`. Its name appears in the **Section plane/face** list on the **Definition** tab.

3 In the **Cut Plot** dialog box, in addition to displaying **Contours**, select the **Vectors** check box.

4 On the **Settings** tab, using the slider set the **Vector spacing** to approximately 0.012.

5 Click **View Settings** in order to specify the parameter whose values will be shown by the contour plot.
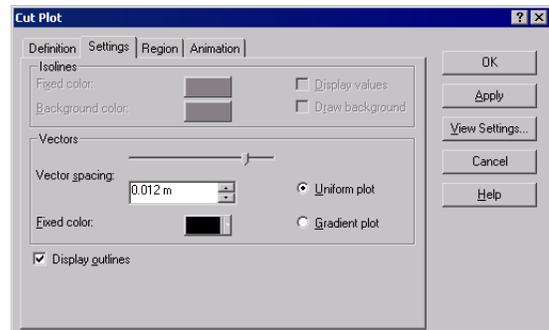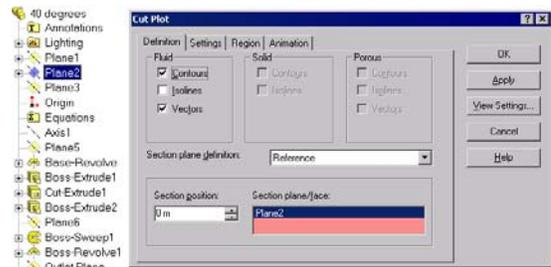
Note: The settings made in the **View Settings** dialog box refer to all **Cut Plots**, **Surface Plots**, **Isosurfaces**, and **Flow Trajectories** specific features. These settings are only applied for the active pane of the SolidWorks graphics area. For example, the contours in all cut and surface plots will show the same physical parameter selected in the **View Settings** dialog box. So, in the **View Settings** dialog box for each of the displaying options (contours, isolines, vectors, flow trajectories, isosurfaces) you specify the displayed physical parameter and the settings required for displaying it through this option. The contour settings can also be applied to **Isolines**, **Vectors**, **Flow Trajectories** and **Isosurfaces**.

---

6 On the **Contours** tab, in the **Parameter** box, select **X-Velocity**.

7 Click **OK** to save changes and exit the **View Settings** dialog box.

8 In the **Cut Plot** dialog box click **OK**. The new **Cut Plot1** item appears in the COSMOSFloWorks design tree.
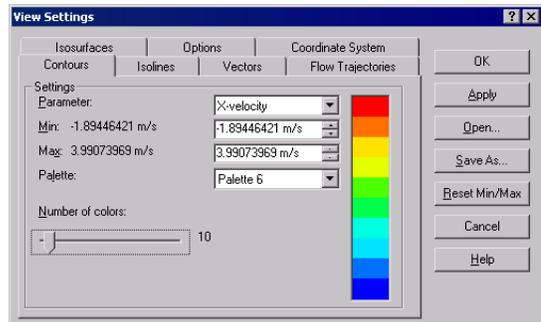
However, the cut plot is not seen through the model. In order to see the plot, you can hide the model by clicking **FloWorks**, **Results**, **Display**, **Geometry**.
Alternatively, you can use the standard SolidWorks **Section View** option.

9 Click **View**, **Display**, **Section View**. Specify `Plane2` as a Section Plane. Click **OK** .
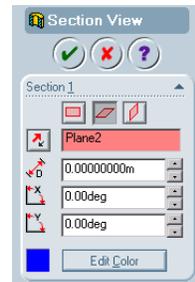
10 In the COSMOSFloWorks design tree, right-click the **Computational Domain** icon and select **Hide**.

Now you can see a contour plot of the velocity and the velocity vectors projected on the plot.
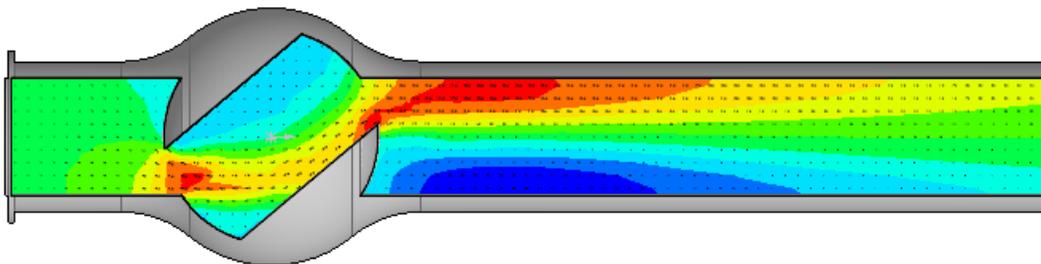
For better visualization of the vortex you can scale small vectors:

11 In the COSMOSFloWorks design tree, right-click the **Results** icon and select **View Settings**.
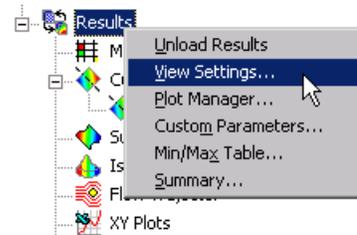
12 In the **View Settings** dialog, click the **Vectors** tab and in the **Arrow size** box type 0.02.
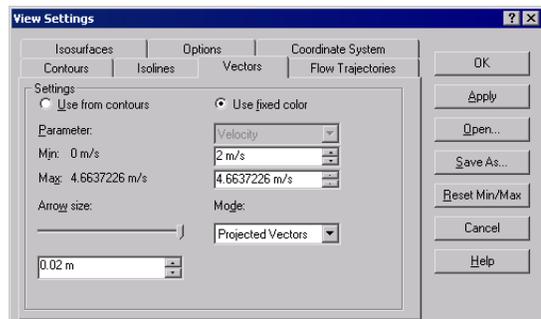
13 Change the **Min** value to 2 m/s.

By specifying the custom **Min** we change the vector length so the vectors whose velocity is less than the specified Min value will have the same length as the

vectors whose velocity is equal to the Min. This allows us to visualize the low velocity area in more details.

14 Click **OK** to save the changes and exit the **View Settings** dialog box. Immediately the cut plot is updated.
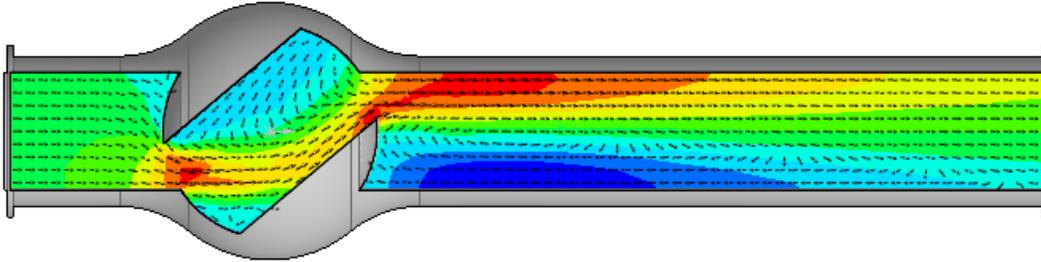
Displaying Flow Trajectories

Using Flow trajectories you can show the flow streamlines and paths of particles with mass and temperature that are inserted into the fluid. Flow trajectories provide a very good image of the 3D fluid flow. You can also see how parameters change along each trajectory by exporting data into Excel. Additionally, you can save trajectories as SolidWorks reference curves.

Right-click the **Cut Plot 1** icon and select **Hide**.

1 Right-click the **Flow Trajectories** icon and select **Insert**. The **Flow Trajectories** dialog box appears.

2 In the COSMOSFloWorks Design Tree, click the **Static Pressure1** item to select the inner face of the **Outlet Lid**. Trajectories launched from the outlet opening will better visualize the vortex occurring downstream the valve's obstacle.
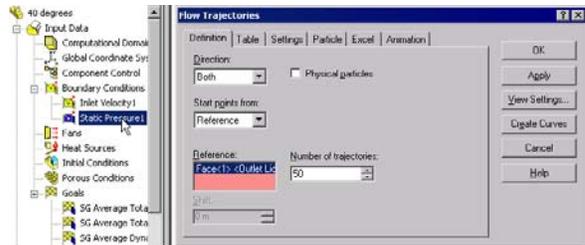
3 Set the **Number of trajectories** to 50.

4 Click the **Setting** tab and decrease the **Maximum length** of trajectories to 2 m.

5 Click **OK** to display trajectories.
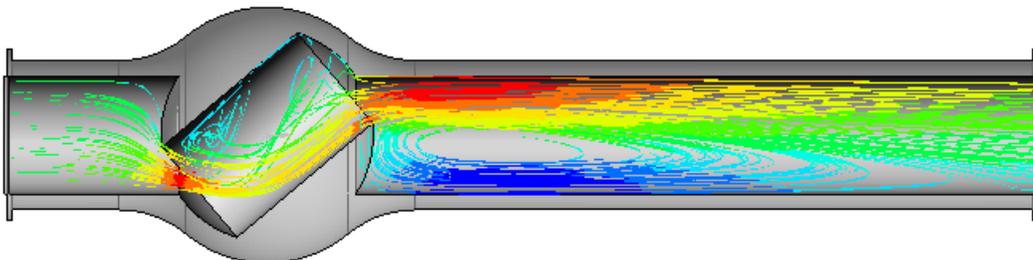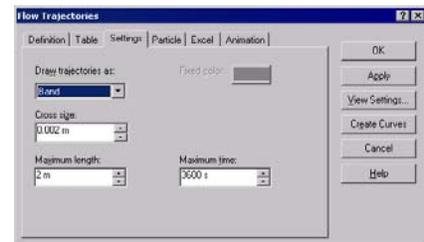
Rotate the model to better view the 3D structure of the vortices.