1

First Steps - Ball Valve Design

This First Steps tutorial covers the flow of water through a ball valve assembly before and after some design changes. The objective is to show how easy fluid flow simulation can be using COSMOSFloWorks and how simple it is to analyze design variations. These two factors make COSMOSFloWorks the perfect tool for engineers who want to test the impact of their design changes.

Conventions

❑   **Bold** refers to buttons, icons or tabs to click on.

❑   `Courier` refers to information you need to enter using the keyboard.

❑   **Menu, Submenu** indicates menu picks found in the main menu.

📖   *Italic is used for helpful notes and explanations.*

Open the SolidWorks Model

1   Click **File**, **Open**. In the **Open** dialog box, browse to the
Ball Valve.SLDASM assembly located in the
First Steps - Ball Valve folder and click **Open** (or dou-
ble-click the assembly). Alternatively, you can drag and
drop the **Ball Valve.SLDASM** file to an empty area of
SolidWorks window. Make sure, that the **default** config-
uration is the active one.

📖   *This is a ball valve. Turning the handle closes or opens
the valve. The mate angle controls the opening angle.*

2   Show the **lids** by clicking the features in the SolidWorks
Feature Manager (Lid 1 and Lid 2).

📖   *We utilize this model for the COSMOSFloWorks simulation without many
significant changes. The user simply closes the interior volume using extrusions we
call lids. In this example the lids are semi-transparent allowing a view into the
valve.*

3   On the **Visualisation** tab, set the **Background** color to white.
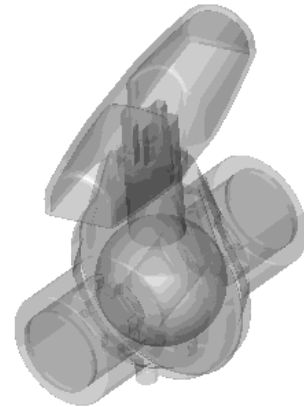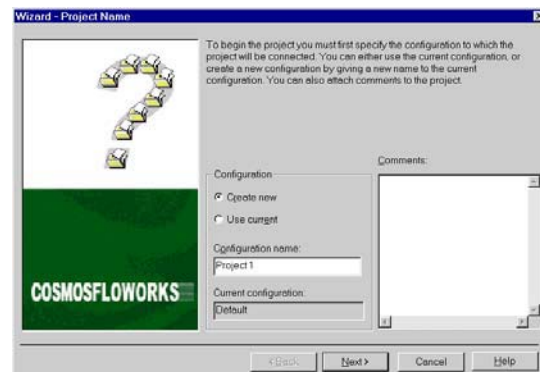
Create a COSMOSFloWorks Project

1   Click **FloWorks**, **Project**, **Wizard**.

2   Once inside the Wizard, select **Create
new** in order to create a new configu-
ration and name it Project 1.

📖   *COSMOSFloWorks will create a new
configuration and store all data in a
new folder.*

Click **Next**.

3   Choose the system of units (**SI** for this project). Please keep in mind that after finishing the Wizard, you can change the unit system anytime with **FloWorks, Units**.
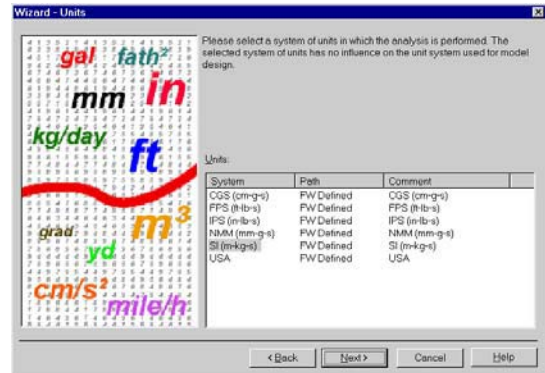
    📖 *Within COSMOSFloWorks, there are some predefined systems of units. You can also define your own and switch between them anytime.*

        Click **Next**.

4   Set the fluid type to **Liquid**. Do not include any other physical features.
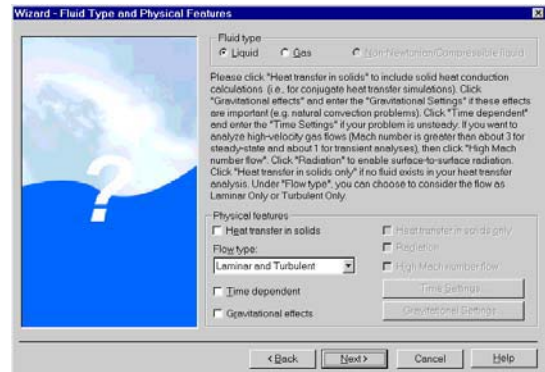
    📖 *COSMOSFloWorks can analyze either incompressible liquids or compressible gases, but not both in the same run. Not only will COSMOSFloWorks calculate the fluid flow, but can also take into account heat transfer within the solid(s) including surface-to-surface radiation. Transient (time dependent) analyses are also possible. Gravitational effects can be included for natural convection cases. The turbulent equations can be removed if the flow is entirely laminar. Finally, COSMOSFloWorks can handle low and high Mach number compressible flows for gases. For this demonstration we will perform a fluid flow simulation using a liquid and will not use any other physical features.*
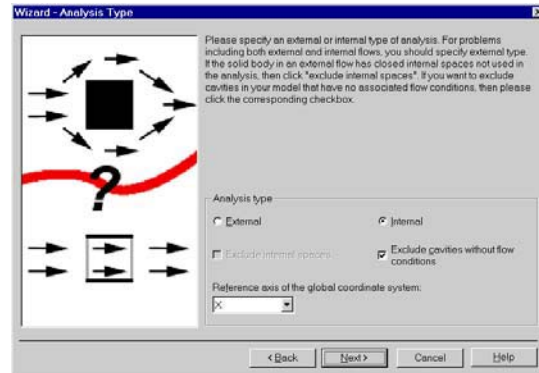
        Click **Next**.

5   Set the analysis type to **Internal**.

   📖 *We want to analyze the flow **through** the structure. This is what we call an internal analysis. The opposite is an external analysis, which is the flow **around** an object. From this dialog box you can also choose to ignore cavities that are not relevant to the flow analysis without having to fill them in using additional features.*
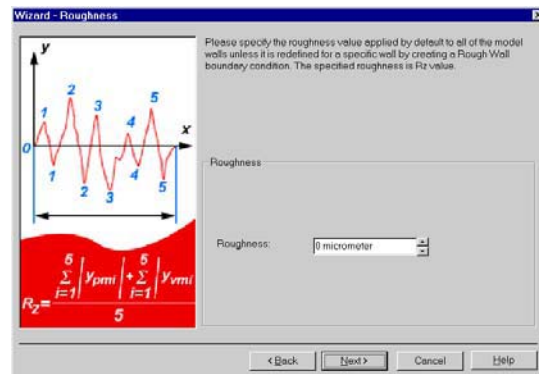
      Click **Next**.

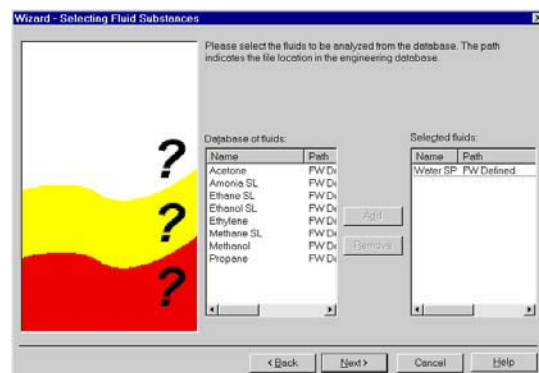**6**   Click **Next** accepting the default zero roughness value for all model walls.

   📖 *You can specify the desired wall roughness value applied by default to all model walls.To set the roughness value for a specific wall you can define a **Real Wall** boundary condition. The specified roughness value is the Rz value.*

7   Choose **Water SP** as the fluid. You can either double-click Water SP or select the item in the left column and click **Add**.

   📖 *COSMOSFloWorks has an integrated database containing several liquids, gases and solids. Solids are used for conduction in conjugate heat transfer analyses. You can easily create your own materials. Up to ten liquids or gases can be chosen for each analysis run.*

      Click **Next**.

8   Click **Next** accepting the default wall conditions.

&   *Since we did not choose to calculate the amount of heat transfer within the solids we have the option of defining a value of heat transfer for the surfaces in contact with the fluid. This box is where we can set the default wall type. Leave the default* **Adiabatic wall** *specifying the walls are perfectly insulated (no heat escapes through the walls).*
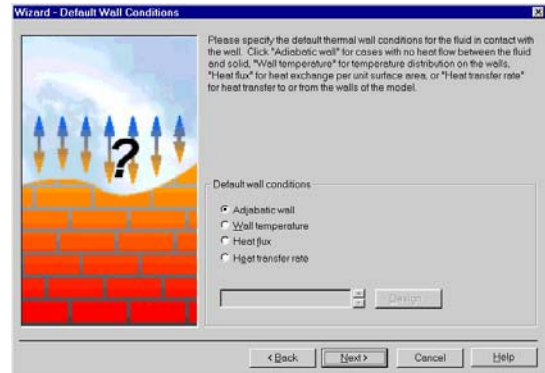
9   Click **Next** accepting the default for the initial conditions.

&   *This box is where we can change the default settings for pressure, temperature and velocity. The closer these values are set to the final values determined in the analysis, the quicker the analysis will finish. Since we do not have any knowledge of the expected final values, we will not modify them for this demonstration.*
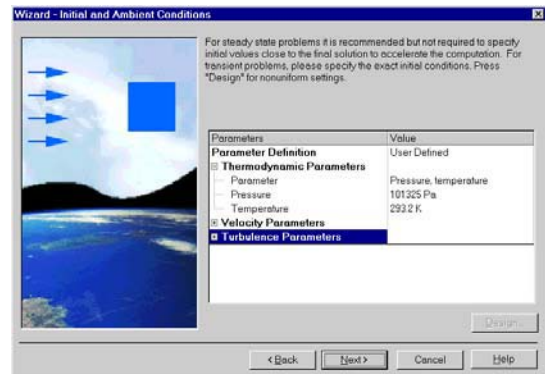
10   Accept the default for the **Result Resolution**.

&   *Result Resolution is a measure of the desired level of accuracy of the results. It controls not only the resolution of the mesh, but also sets many parameters for the solver, e.g. the convergence criteria. The higher the Result Resolution, the finer the mesh will be and the stricter the convergence criteria will be set. Thus, Result Resolution determines the balance between precise results and computation time. Entering values for the minimum gap size and minimum wall thickness is important when you have small features.*
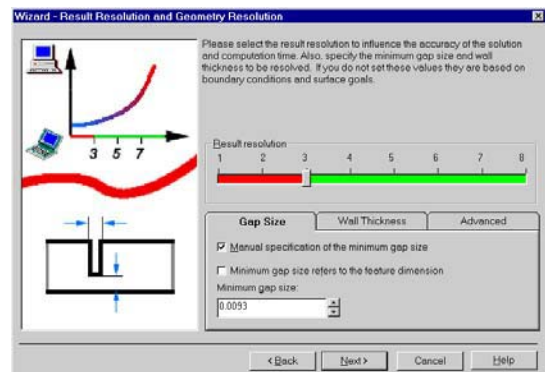
Setting these values accurately ensures your small features are not "passed over" by the mesh. For our model we type the value of the minimum flow passage as the minimum gap size.

Click the **Manual specification of the minimum gap size** box. Enter the value 0.0093 m for the minimum flow passage.

Click **Next**.

11   Click **Finish**.

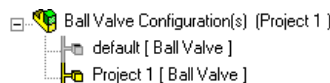   📖 *This box provides an overview of the selections made during the wizard.*

Now COSMOSFloWorks creates a new configuration with the COSMOSFloWorks data attached.
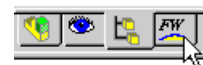
Click on the SolidWorks **Configuration Manager** to show the new configuration.
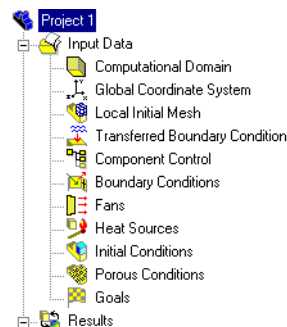
   📖 *Notice the name of the new configuration has the name you entered in the Wizard.*

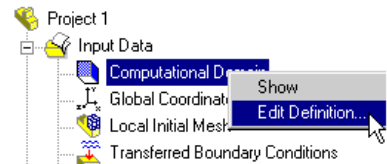Go to the **COSMOSFloWorks Design Tree** and open all the icons.

   📖 *We will use the COSMOSFloWorks Design Tree to define our analysis, just as the SolidWorks Feature Manager is used to design your models.*

Right-click the **Computational Domain** icon and select **Hide** to hide the black wire-frame box.

📖 *Computational Domain is the icon used to modify the size and visualization of the volume being analyzed. The wireframe box enveloping the model is the visualization of the limits of the computational domain.*
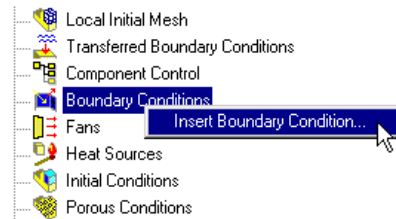
Boundary Conditions

A **boundary condition** is required anywhere fluid enters or exits the system and can be set as a Pressure, Mass Flow, Volume Flow or Velocity.

1   In the COSMOSFloWorks Design Tree, right-click the **Boundary Conditions** icon and select **Insert Boundary Condition**.
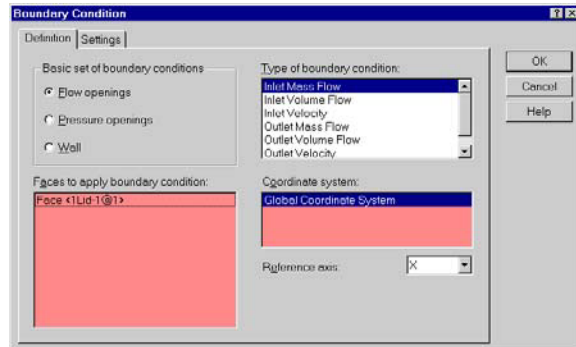
2   Select the **inner** face of the **Lid 1** part as shown. (To access the inner face, right-click the mouse to cycle through the faces under the cursor until the inner face is highlighted, then click the left mouse button).
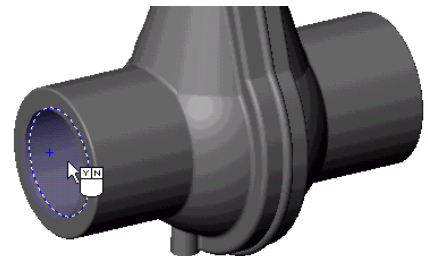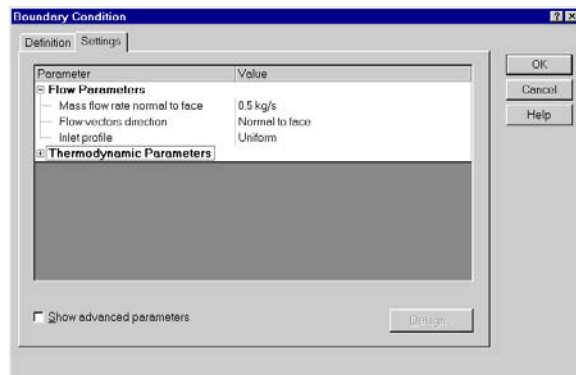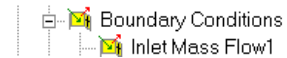
3   Select **Flow openings** and **Inlet Mass Flow**.

4   Set the **Mass flow rate normal to face** to 0.5 kg/s under the **Settings** tab.

5   Click **OK**. The new **Inlet Mass Flow1** item appears in the COSMOSFloWorks Design Tree.
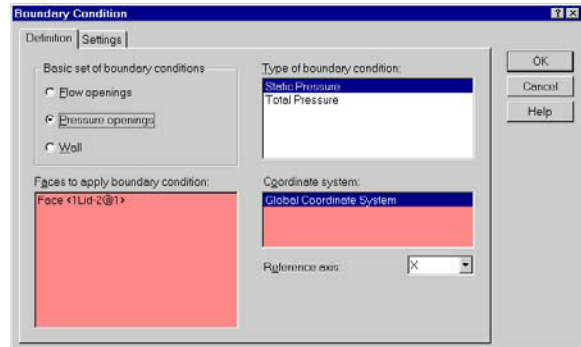

    With the definition just made, we told COSMOSFloWorks that at this opening 0.5 kilogram of water per second is flowing into the valve. Within this dialog box we can also specify a swirl to the flow, a non-uniform profile and time dependent properties to the flow. The mass flow at the outlet does not need to be specified due to the conservation of mass; mass flow in equals mass flow out. Therefore another different condition must be specified. An outlet pressure should be used to identify this condition.

6   Select the **inner** face of the **Lid 2** part as shown. (To access the inner face, right-click the mouse to cycle through the faces under the cursor until the inner face is highlighted, then click the left mouse button).

7   Right-click the **Boundary Conditions** icon and select **Insert Boundary Condition**.
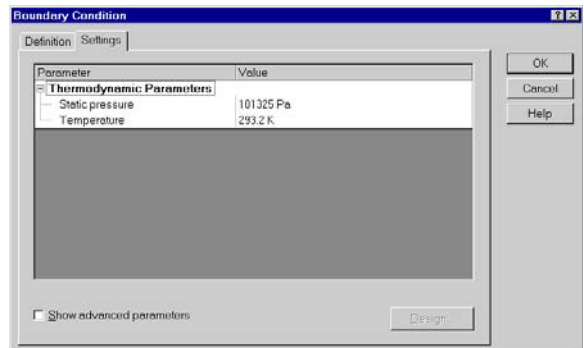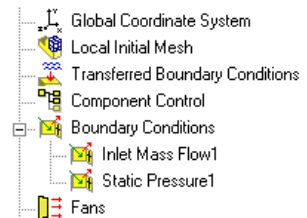
8   Select **Pressure openings** and **Static Pressure**.

**9**   Keep the defaults under **Settings**.

10   Click **OK**. The new **Static Pressure1** item appears in the COSMOSFloWorks Design Tree.

     With the definition just made, we told COSMOSFloWorks that at this opening the fluid exits the model to an area of static atmospheric pressure. Within this dialog box we can also set time dependent properties to the pressure.
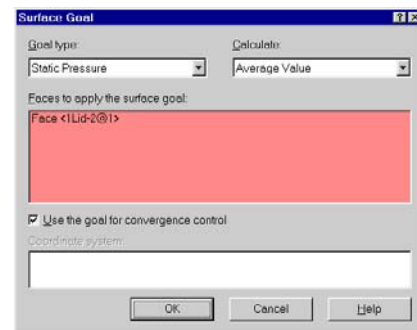
Define the Engineering Goal

1   Right-click the COSMOSFloWorks Design Tree
        Goals icon and select **Insert Surface Goal**.
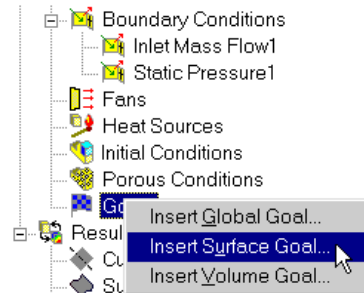
2   Click the **Inlet Mass Flow1** item to select the face
        where it is going to be applied

3   Select **Static Pressure** as **Goal type**.

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

📖   *If the Use the goal for convergence control
        check box is not selected for a goal, it will not
        influence the task stopping criteria. Such goals
        can be used as monitoring parameters to give
        you additional information about processes
        occurring in your model without affecting the
        other results and the total calculation time.*

5   Click **OK**. The new **SG Average Static Pressure1**
        item appears in the COSMOSFloWorks Design
        Tree.

📖   *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.
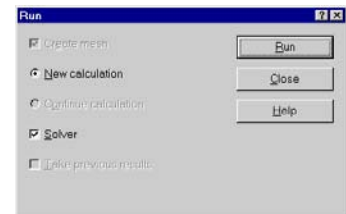
Click **File, Save**.

Solution

**1**   Click **FloWorks**, **Solve**, **Run**.

2   Click **Run**.

📖   *The solver should take a few minutes to run on a 500MHz PIII platform.*

Monitor 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. Do not be surprised when the error message "A vortex crosses the pressure opening" is listed. We will explain this later during the demonstration.*
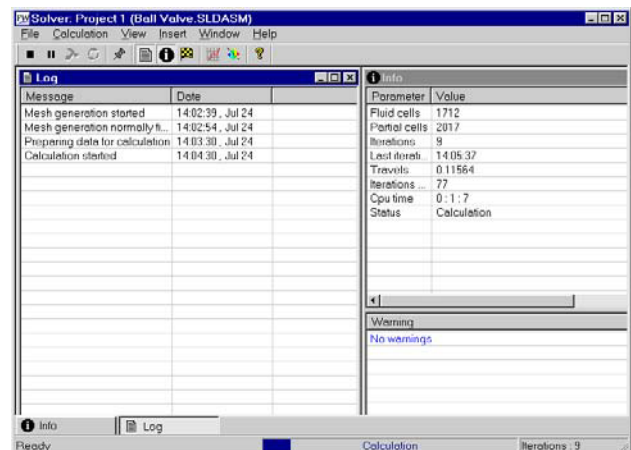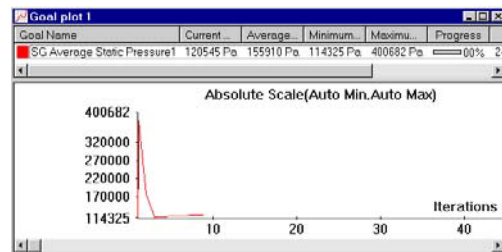
1   Click **Insert Goal Plot** 📈 on the **Solver** toolbar. The **Add/Remove Goals** dialog box appears.

2   Double-click the **SG Average Static Pressure1** in the **Available goals** list and click **OK**.
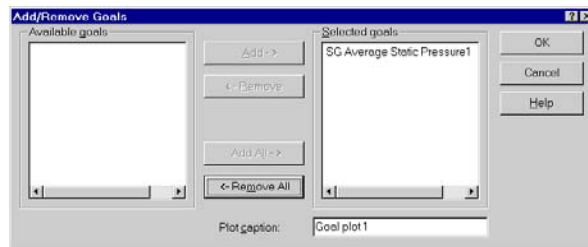
     This is the Goals Dialog box and each goal created earlier is listed above. Here you can see the current value and graph for each goal as well as the current progress towards completion given as a percentage. The progress value is only an estimate and the rate of progress generally increases with time.

3   Click **Insert Preview** on the **Solver** toolbar.

4   This is the **Preview Settings** dialog box. Selecting any SolidWorks plane from the SolidWorks Feature Manager Tree and pressing **OK** will create a preview plot of the solution on that plane. For this model `Plane2` is a good choice to use as the preview plane. The preview plane can be chosen anytime from the SolidWorks Feature Manager.
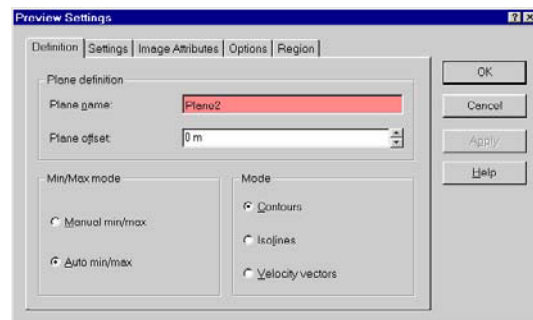
     The preview allows one to look at the results while the calculation is still running. This helps to determine if all the boundary conditions are correctly defined and gives the user an idea of how the solution will look even at this early stage. At the start of the run the results might look odd or change 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.
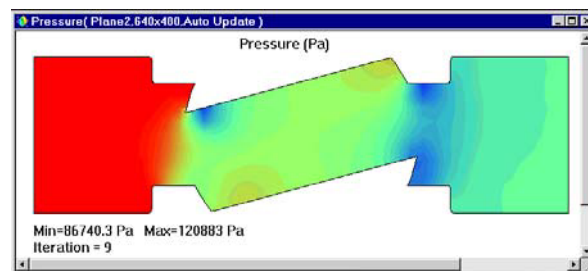
5   When the solver is finished, close the monitor by clicking **File**, **Close**.

Access the Results

1   Click **FloWorks**, **Results**, **Display**, **Transparency** and set the model transparency to 0.75.
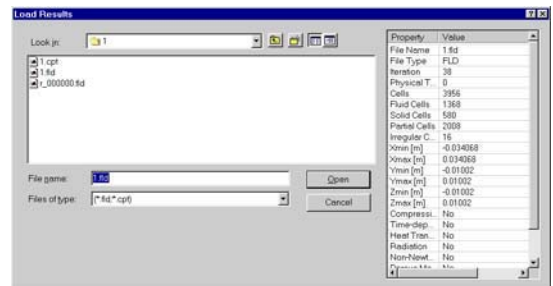
     📖   *The first step for results is to generate a transparent view of the geometry, a 'glass-body'. This way, you can easily see where cut planes etc. are located with respect to the geometry.*

2   Right-click the **Results** icon and select **Load Results** to activate the postprocessor.
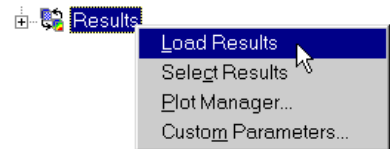
3   In the **Load Results** dialog box, keep the default project's results file (1.fld) and click **Open**.
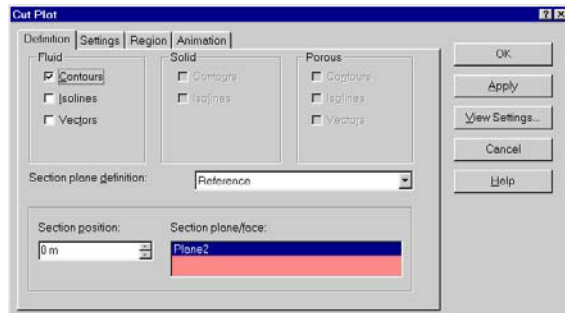
Cut Plots

1   Right-click the **Cut Plots** icon and select **Insert**.

2   Specify a plane. Choose **Plane 2** as the cut plane. To do this, click on the SolidWorks Feature Manager tab and select **Plane 2**.

3   Click **OK**.

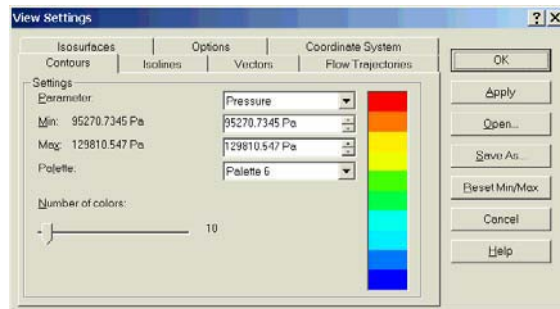This is the plot you should see.

📖   *A cut plot displays any result on any SolidWorks plane. The representation can be as a contour plot, as isolines or as vectors – and also in any combination of the above (e.g. contour with overlaid vectors.*

4   To access additional options for this and other plots, either double-click on the color scale or right-click the **Results** icon and select **View Settings**.
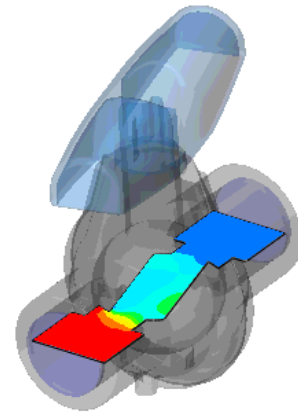
📖   *Within the View Settings dialog box you have the ability to change the global options for each plot type. Some options available are: changing the variable being displayed and the number of colors used for the scale. The best way to learn each of these options is through experimentation.*

5   Change the contour cut plot to a vector cut plot. To do this, right-click the **Cut Plot 1** icon and select **Edit Definition.**
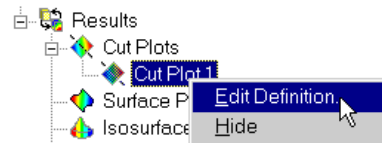
6 Clear **Contours** and select **Vectors** in the plot definition.

7 Click **OK**.
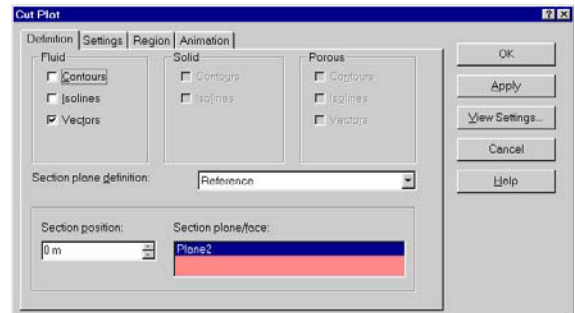
This is the plot you should see.

  The vectors can be made larger from the Vectors tab in the View Setting dialog box. The vector spacing can also be controlled from the Settings tab in the Cut Plot dialog box. Notice how the flow must navigate around the sharp corners on the Ball. Our design change will focus on this feature.
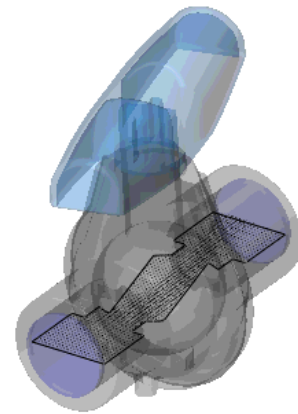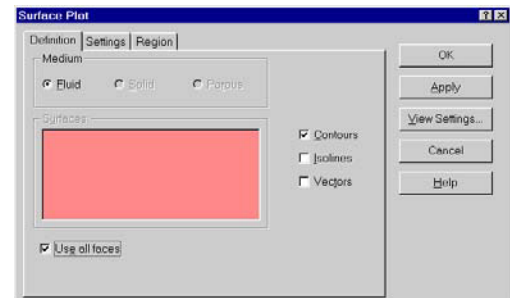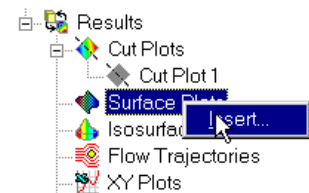
Surface Plots

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

1 Right-click the **Surface Plots** icon and select **Insert**.

2 Select the **Use all faces** check box.

3 Select the **Contours** check box.

  The same basic options are available for Surface Plots as for Cut Plots. Feel free to experiment with different combinations on your own.

4   Click **OK** and you get the following picture:

 📖 *This plot shows the pressure distribution on all faces of
the valve in contact with the fluid. You can also select
one or more single surfaces for this plot, which do not
have to be planar.*
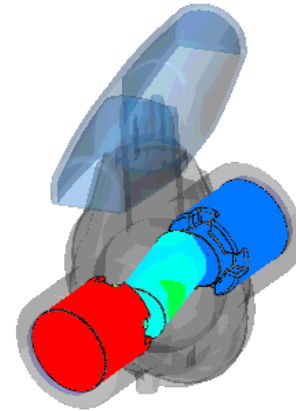
Isosurface Plots

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

1   Right-click the **Isosurfaces** icon and select **Show**.

This is the plot that will appear.

 📖 *The Isosurface is a 3-Dimensional surface created by
COSMOSFloWorks at a constant value for a specific
variable. The value and variable can be altered in the
**View Settings** dialog box under the **Isosurfaces** tab.*

2   Right-click in the white area and select **View Settings** to
enter the dialog.

3   Go to **Isosurfaces** tab.

4   Examine the options under this dialog
box. Try making two changes. The first
is to click in the **Use from contours** so
that the color of the isosurface be col-
ored in the same manner as the pressure
value on a contour plot.
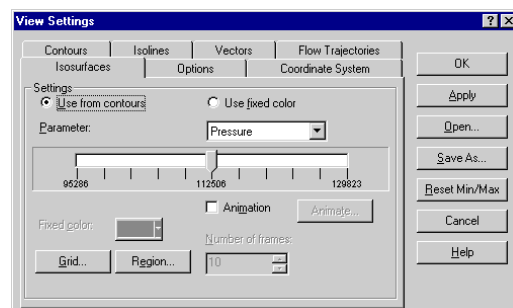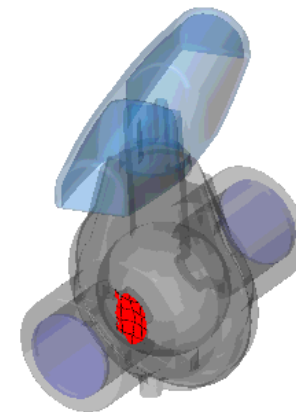
5   Secondly, click at a second location on the slide bar and notice the addition of a second slider. These sliders can later be removed by dragging them all of the way out of the dialog box.

6   Click **OK** and you should see something similar to this image.
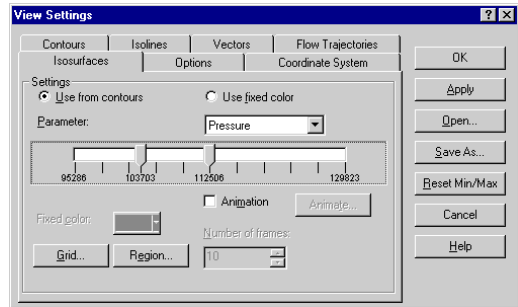
📖   *The isosurface is a useful way of determining the exact area, in 3-dimensions, where the flow reached a certain pressure, velocity or other variable.*
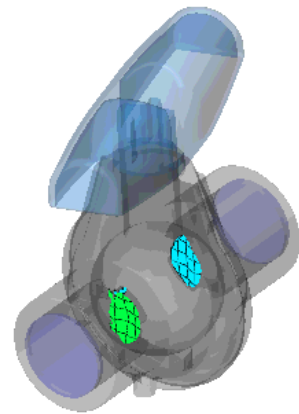
Flow Trajectory Plots

Right-click the **Isosurfaces** icon and select **Hide**.

1   Right-click the **Flow Trajectories** icon and select **Insert**.
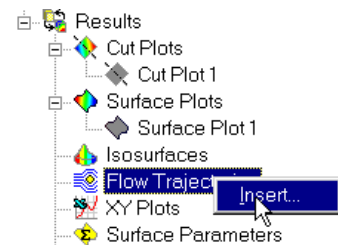
2   In the COSMOS-FloWorks Design Tree, click the **Static Pressure1** item to select the **inner** face of the outlet **Lid 2** part.

3   Set the **Number of trajectories** to 16.

4   Click **OK** and your picture should look like the following:

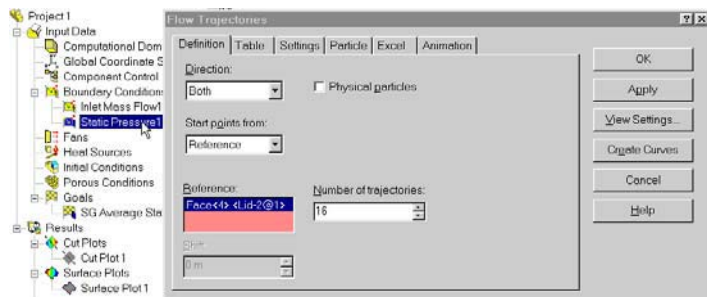   📖   *Using Flow trajectories you can show the flow streamlines and paths of particles with mass and temperature that are inserted into the fluid. Flow trajectories provide a very good image of the 3D fluid flow. You can also see how parameters change along each trajectory by exporting data into Excel. Additionally, you can save trajectories as SolidWorks reference curves.*
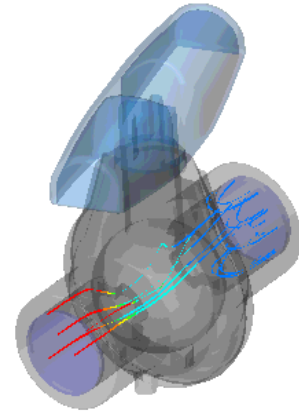
    📖   *For this plot we selected the outlet lid (any flat face or sketch can be selected) and therefore every trajectory crosses that selected face. The trajectories can also be colored by values of whatever variable chosen in the **View Settings** dialog box. Notice the trajectories that are entering and exiting through the exit lid. This is the reason for the warning we received during the solver. COSMOSFloWorks warns us of inappropriate analysis conditions so that we do not need to be CFD experts. When flow both enters and exits the same opening, the accuracy of the results will worsen. In a case such as this, one would typically add the next component to the model (such as a pipe extending the computational domain) so that the vortex does not occur at an opening.*

XY Plots

Right-click the **Flow Trajectories 1** icon and select **Hide**.

We want to plot pressure and velocity along the valve. We have already created a SolidWorks sketch containing several lines.

This sketch work does not have to be done ahead of time and your sketch lines can be created after the analysis has finished. Take a look at Sketch 1 in the SolidWorks Feature Manager Tree.

**1**   Right-click the **XY Plots** icon and select **Insert**.
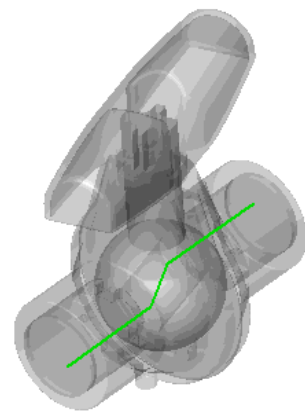
2   Choose **Velocity** and **Pressure** as **Physical parameters**. Select **Sketch1** from the SolidWorks Feature Manager.

Leave all other options as defaults.
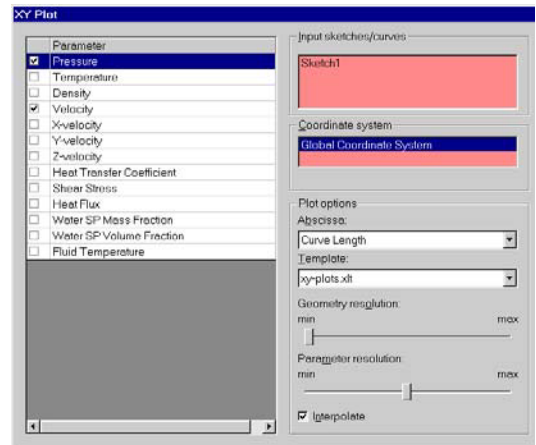
3   Click **OK**. MS Excel will open and generate two lists of data points as well as two graphs, one for Velocity and the other for Pressure. One of these plots is the one shown below. You will need to toggle between different sheets in Excel to view each graph.
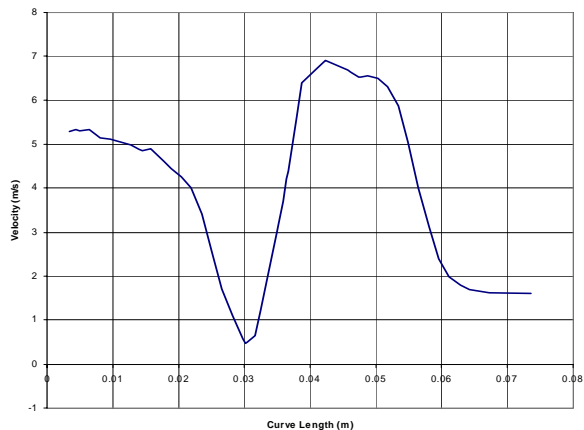
   *The XY Plot allows you to view any result along sketched lines. The data is put directly into Excel.*

Surface Parameters

Surface Parameters is the icon used to determine pressures, forces, heat flux as well as many other variables on any face within your model contacting the fluid. For this type of analysis it would probably be of interest to calculate the average static pressure drop from the valve inlet to outlet.

1 Right-click the **Surface Parameters** icon and select **Insert**.

2 In the COSMOSFloWorks Design Tree, click the **Inlet Mass Flow1** item to select the **inner** face of the inlet **Lid 1** part.

3 Click **Apply**.

4 Select the **Local** tab.
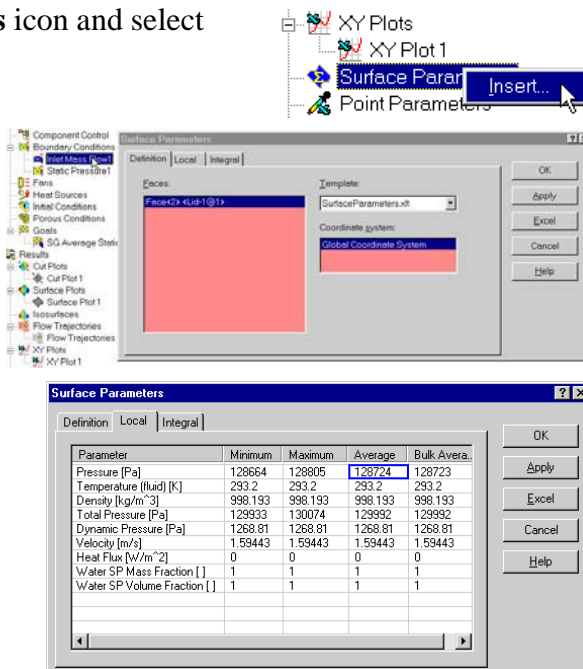
 📖 *The average static pressure at the inlet face is shown to be 128724 Pa. We already know that the outlet static pressure is 101325 Pa since we applied it previously as a boundary condition. So, the average static pressure drop through the valve is calculated to be 27399 Pa.*

5 Close the **Surface Parameters** dialog box.

## Analyze a Design Variant in the SolidWorks Ball part

 📖 *This section is intended to show you how easy it is to analyze design variations. The variations can be different geometric dimensions, new features, new parts in an assembly – whatever! This is the heart of COSMOSFloWorks and is what allows design engineers to quickly and easily determine which designs have promise, and which designs are unlikely to be successful. For this example, we will see how filleting two sharp edges will influence the pressure drop through the valve. If there is no improvement, it will not be worth the extra manufacturing costs.*

Create a new configuration using the SolidWorks Configuration Manager Tree.

1   Right-click the root item in the SolidWorks Configuration
Manager and select **Add Configuration**.

2   In the **Configuration Name** box type
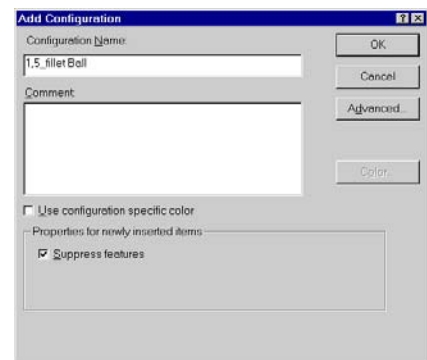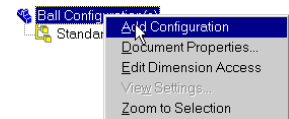`Project 2.`

3   Click **OK**.
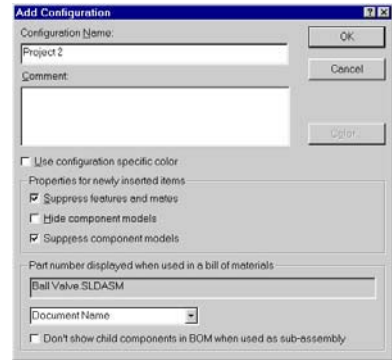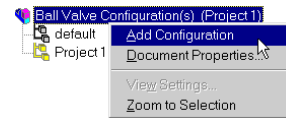
4   Go to SolidWorks FeatureManager, right-click the
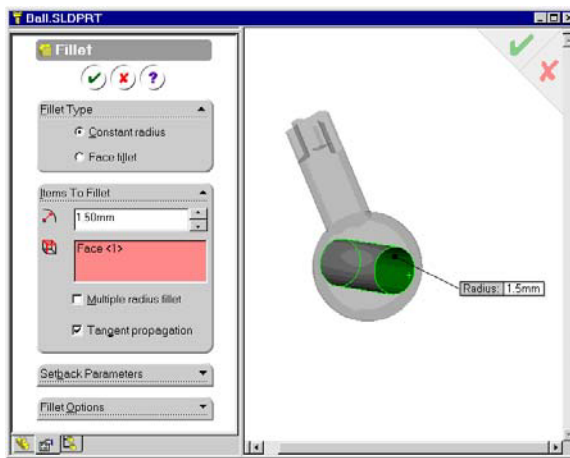**Ball** item and select **Open ball.sldprt**.
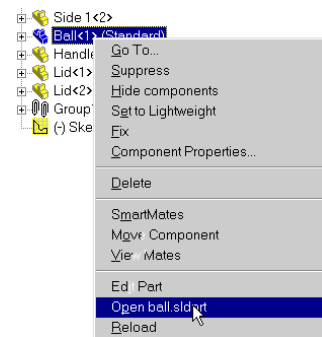
Create a new configuration using the SolidWorks Configuration Manager Tree.

1   Right-click the root item in the SolidWorks Configuration
Manager and select **Add Configuration**.

2   Name the new configuration as
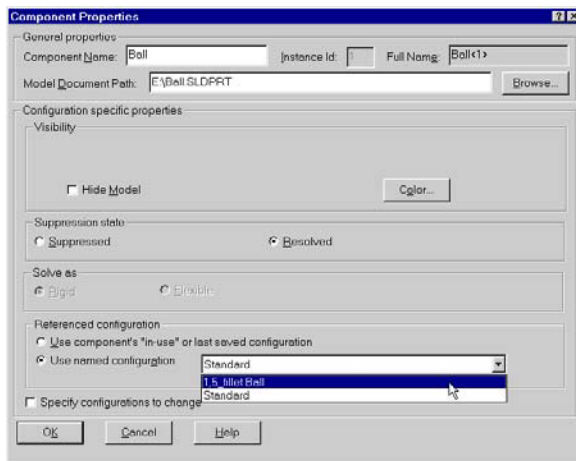`1,5_fillet Ball.`

3   Click **OK**.

4 Add a 1,5 mm fillet to the shown face.

5 Back in the assembly, right-click the **Ball** item and select **Component Properties**.

6 At the bottom of the **Component Properties** dialog box change the configuration of the Ball part to the new filleted one.

**7** Click **OK** to confirm and close the dialog.

Now we have replaced the old ball with our new 1.5_fillet Ball. All we need to do now is re-solve the assembly and compare the results of the two designs. In order to make the results comparable with the previous model, it would be necessary to adjust the valve angle to match the size of the flow passage of the first model. In this example, we will not do this.
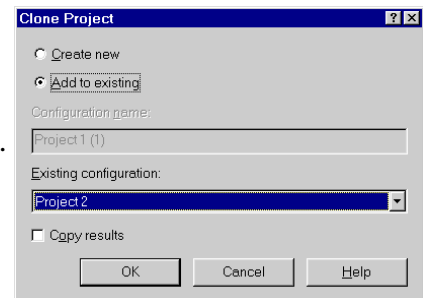
8   Activate **Project 1** by using the Configuration Manager Tree.
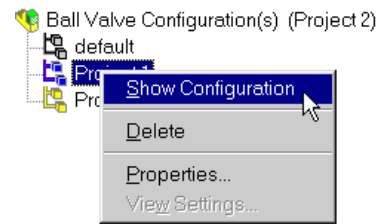
Clone the Project

1   Click **FloWorks**, **Project**, **Clone Project**.

2   Click **Add to existing**.

3   In the **Existing configuration** list select **Project 2**.

    Click **OK**. Select **Yes** for each message dialog box that appears after you click **OK**.

    Now the COSMOSFloWorks project we have chosen is added to the SolidWorks project which contains the geometry that has been changed. All our input data are copied, so we do not need to define our openings or goals again. The Boundary Conditions can be changed, deleted or added. All changes to the geometry will only be applied to this new configuration, so the old results are still saved. Please follow the previously described steps for solving and for viewing the results.

## Analyze a Design Variant in the COSMOSFloWorks Application

    In the previous sections we examined how you could compare results from different geometries. You may also want to run the same geometry over a range of flow rates. This section shows how quick and easy it can be to do that kind of parametric study. Here we are going to change the mass flow to 0.75 kg/s.

Activate the **Project 1** configuration.

1 Create a copy of the **Project 1** project by clicking **FloWorks**, **Project**, **Clone Project**.

2 Type `Project 3` for the new project name and click **OK**.
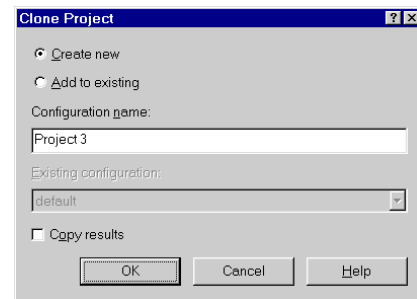
COSMOSFloWorks now creates a new configuration. All our input data are copied, so we do not need to define our openings or goals again. The Boundary Conditions can be changed, deleted or added. All changes to the geometry will only be applied to this new configuration, so the old results remain valid. After changing the inlet flow rate value to 0.75 kg/s you would be ready to run again. Please follow the previously described steps for solving and for viewing the results.

Imagine being the designer of this ball valve. How would you make decisions concerning your design? If you had to determine whether the benefit of modifying the design as we have just done outweighed the extra costs, how would you do this? Engineers have to make decisions such as this every day, and COSMOSFloWorks is a tool to help them make those decisions. Every engineer who is required to make design decisions involving fluid and heat transfer should utilize COSMOSFloWorks to test their ideas, allowing for fewer prototypes and quicker design cycles.