1

First Steps - Ball Valve Design

This First Steps tutorial deals with 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 with Flow Simulation and how simple it is to analyze design variations. These two factors make Flow Simulation the perfect tool for engineers who want to test the impact of their design changes.

Open the SolidWorks Model

1 Copy the **First Steps - Ball Valve** folder from the installation directory into your working directory and ensure that the files are not read-only since Flow Simulation will save input data to these files. Run **Flow Simulation**.

2 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 double-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** configuration is the active one.

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

3 Highlight the **lids** by clicking the features in the FeatureManager design tree (Lid <1> and Lid <2>).

📖 *We utilize this model for the Flow Simulation simulation without any significant changes. The user simply closes the interior volume using extrusions that we call lids. In this example the lids are made semi-transparent so you may look into the valve.*
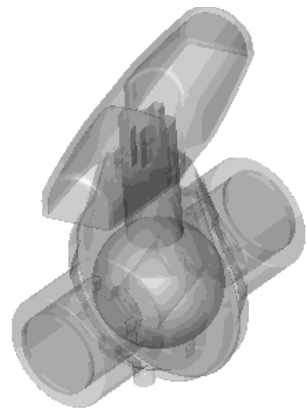
Create a Flow Simulation Project

1   In the main menu click **Flow Simulation**, **Project**, **Wizard**.

2   Once inside the Wizard, select **Create new** in order to create a new configuration and name it Project 1.
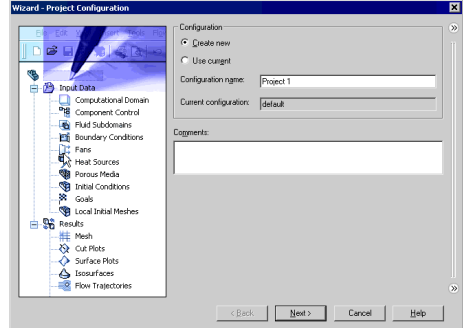
&   *Flow Simulation will create a new configuration and store all data in a new folder.*

   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 at any time by clicking **Flow Simulation**, **Units**.
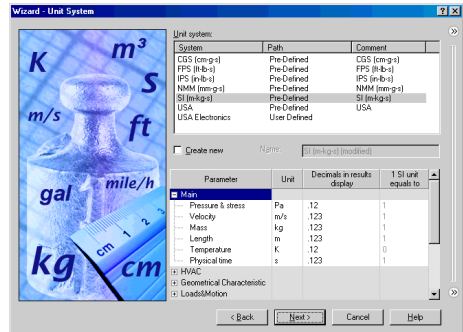
&   *Within Flow Simulation, there are several predefined systems of units. You can also define your own and switch between them at any time.*

   Click **Next**.

4   Keep the default **Internal** analysis type. Do not include any physical features.
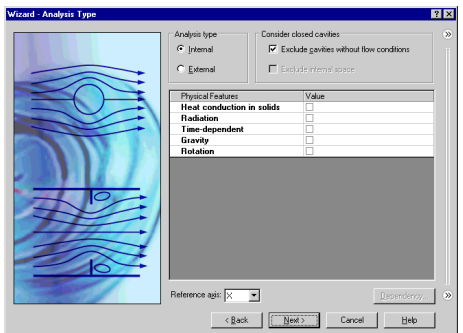
&   *We want to analyze the flow **through** the structure. This is what we call an internal analysis. The alternative is an external analysis, which is the flow **around** an object. In this dialog box you can also choose to ignore cavities that are not relevant to the flow analysis, so that Flow Simulation will not waste memory and CPU resources to take them into account.*

&   *Not only will Flow Simulation calculate the fluid flow, but can also take into account heat conduction within the solid, including surface-to-surface radiation. Transient (time-dependent) analyses are also possible. Gravitational effects can be included for natural convection cases. Analysis of rotating equipment is one more option available. We skip all these features, as none of them is needed in this simple example.*

   Click **Next**.

5 In the **Fluids** tree expand the **Liquids** item and choose **Water** as the fluid. You can either double-click **Water** or select the item in the tree and click **Add**.

📖 *Flow Simulation is capable of calculating flow of fluids of different types in the same analysis, but fluids of different types must be separated by walls. A mixing of fluids may be considered only if the fluids are of the same type.*
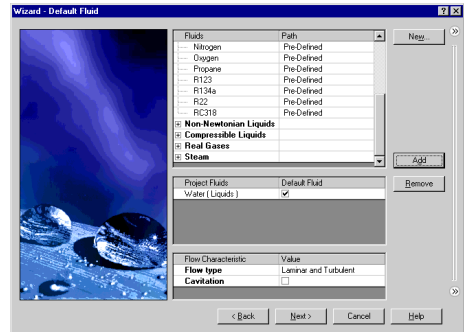
📖 *Flow Simulation has an integrated database containing properties of several liquids, gases and solids. Solids are used in conjugate heat conduction analyses. You can easily create your own materials. Up to ten liquids or gases can be chosen for each analysis run.*

📖 *Flow Simulation can analyze any flow type: Turbulent only, Laminar only or Laminar and Turbulent. The turbulent equations can be disregarded if the flow is entirely laminar. Flow Simulation can also 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 keep the default flow characteristics.*

Click **Next**.

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

📖 *Since we did not choose to consider heat conduction in solids, we have an option to define a value of heat transfer for all surfaces of the model being in contact with the fluid. Keep the default **Adiabatic wall** to specify that the walls are perfectly insulated.*
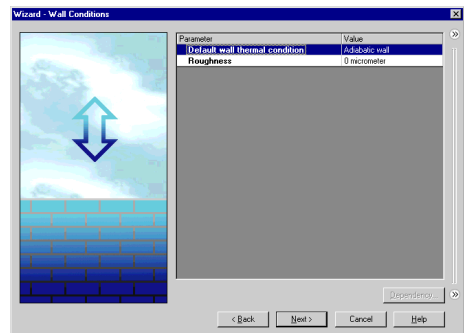
📖 *You can also specify a wall roughness value applied by default to all model walls. The specified roughness value is the R_z value.*

📖 *To set a heat transfer or roughness value for a specific wall, you can define a **Real Wall** boundary condition.*

7   Click **Next** accepting the default for the initial conditions.

  📖 *On this step we can change the default settings for pressure, temperature and velocity. The closer these values 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.*
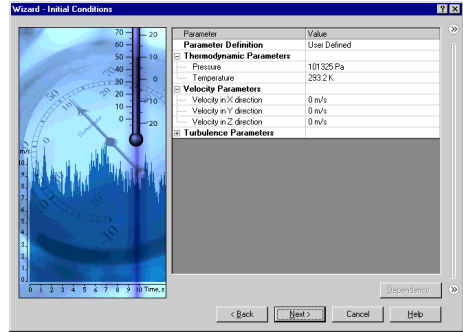
8   Accept the default for the **Result Resolution**.
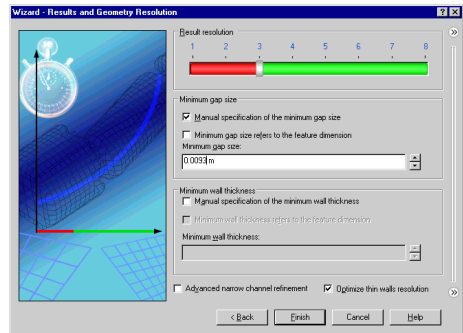
  📖 *Result Resolution is a measure of the desired level of accuracy of the results. It controls not only the resolution of the geometry by the mesh, but also sets many parameters for the solver, e.g. convergence criteria. The higher the Result Resolution, the finer the mesh will be and the stricter convergence criteria will be set. Thus, Result Resolution determines the balance between results precision and computation time. Entering values for the minimum gap size and minimum wall thickness is important when you have small features. Accurately setting these values ensures that the small features of the model will not be "passed over" by the mesh. For our model we type the value of the minimum flow passage as the minimum gap size.*

Select the **Manual specification of the minimum gap size** check box. Type the value of 0.0093 m for the **Minimum gap size**.
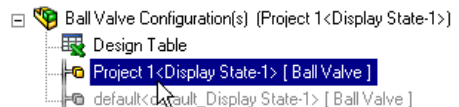
Click **Finish**.

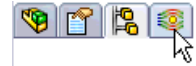Now Flow Simulation creates a new configuration with the Flow Simulation data attached.

Click on the **Configuration Manager** to show the new configuration.

  📖 *Notice that the new configuration has the name that you entered in the Wizard.*

Go to the **Flow Simulation Analysis Tree** and expand all the items in the tree.
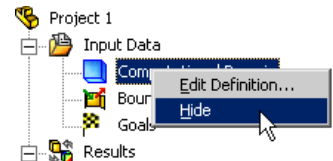
📖 *We will use the Flow Simulation Analysis Tree to define our analysis, just as you use the FeatureManager design tree to design your models. The Flow Simulation analysis tree is fully customizable; anytime you can select which folders are shown and which folders are hidden. A hidden folder becomes visible when you add a new feature of the corresponding type. The folder remains visible until the last feature of this type is deleted.*
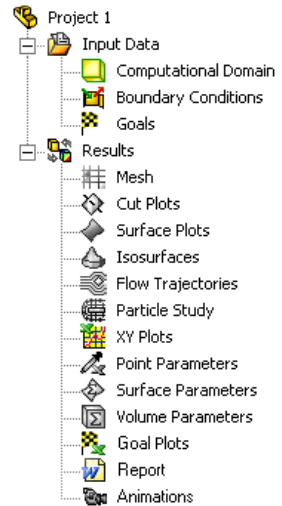
Right-click the **Computational Domain** icon and select **Hide** to hide the wireframe box.

📖 *The Computational Domain icon is 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 where fluid enters or exits the model and can be specified as a Pressure, Mass Flow Rate, Volume Flow Rate or Velocity.
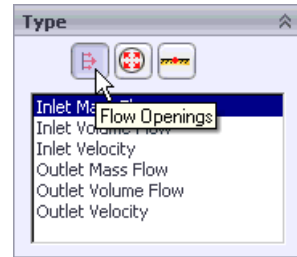
1 In the Flow Simulation Analysis Tree, right-click the **Boundary Conditions** icon and select **Insert Boundary Condition**.
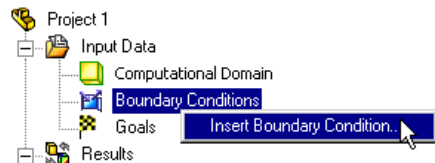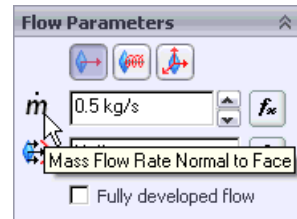
2   Select the **inner** face of the **Lid <1>** part as shown. (To access the inner face, right-click the **Lid <1>** in the graphics area and choose **Select Other** 🔲, move the mouse pointer over items in the list until the inner face is highlighted, then click the left mouse button).
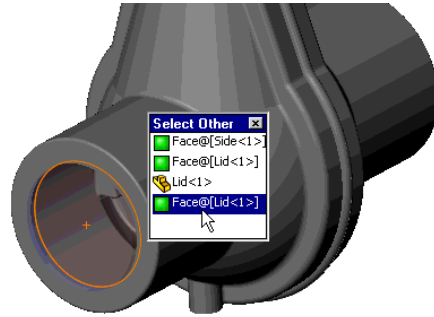
3   Select **Flow Openings** 📥 and **Inlet Mass Flow**.

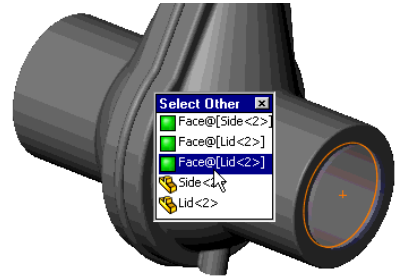**4**   Set the **Mass Flow Rate Normal to Face** $\dot{m}$ to 0.5 kg/s.

5   Click **OK** ✔. The new **Inlet Mass Flow 1** item appears in the Flow Simulation Analysis tree.
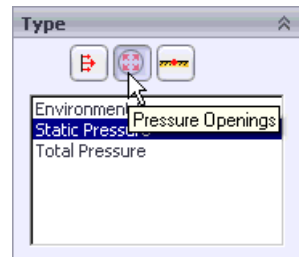
📖 *With the definition just made, we told Flow Simulation that at this opening 0.5 kilogram of water per second is flowing into the valve. Within this dialog we can also specify swirling of the flow, a non-uniform profile and time-dependent properties of the flow. The mass flow rate at the outlet does not need to be specified due to the conservation of mass; inlet mass flow rate equals outlet mass flow rate. Therefore, a different condition must be specified, such as outlet pressure.*

6 Select the **inner** face of the **Lid <2>** part as shown. (To access the inner face, right-click the **Lid <2>** in the graphics area and choose **Select Other** 🔲, move the pointer over items in the list until the inner face is highlighted, then click the left mouse button).

7 In the Flow Simulation Analysis Tree, right-click the **Boundary Conditions** icon and select **Insert Boundary Condition**.

8 Select **Pressure Openings** 🟠 and **Static Pressure**.

9 Keep the defaults under **Thermodynamic Parameters**, **Turbulence Parameters**, **Boundary Layer** and **Options**.

10 Click **OK** ✔ . The new **Static Pressure 1** item appears in the Flow Simulation Analysis tree.
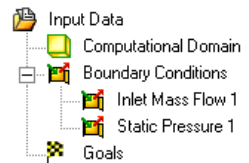
📖 *With the definition just made, we told Flow Simulation that at this opening the fluid exits the model to an area of static atmospheric pressure. Within this dialog we can also set a time-dependent properties pressure.*

Define the Engineering Goal

1 Right-click the **Goals** icon in the Flow Simulation Analysis Tree and select **Insert Surface Goals**.
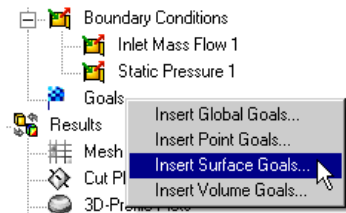
2   Click the **Flow Simulation Analysis Tree** tab and click the **Inlet Mass Flow 1** item to select the face where the goal is going to be applied.
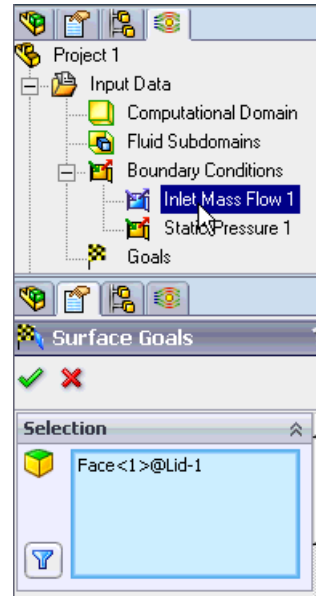
3   In the **Parameter** table, select the **Av** check box in the **Static Pressure** row. The already selected **Use for Conv.** check box means that the created goal will be used for convergence control.

📖 *If the* **Use for Conv.** *(Use for Convergence Control) check box is not selected, the goal will not influence the calculation stopping criteria. Such goals can be used as monitoring parameters to give you additional information about processes in your model without influencing the other results and the total calculation time.*

4   Click **OK** ✔ . The new **SG Av Static Pressure 1** item appears in the Flow Simulation Analysis tree.
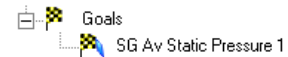
📖 *Engineering goals are the parameters of interest. Setting goals is a way of conveying to Flow Simulation what you are trying to get out of the analysis, as well as a way to reduce the time Flow Simulation needs to reach a solution. By setting a parameter as a project goal you give Flow Simulation information about parameters that are important to converge upon (the parameters selected as goals) and parameters that can be computed with less accuracy (the parameters not selected as goals) in the interest of the calculation time. Goals can be set throughout the entire domain (Global Goals), within a selected volume (Volume Goals), for a selected surface area (Surface Goals), or at given point (Point Goals). Furthermore, Flow Simulation can consider the average value, the minimum value or the maximum value of the goal. You can also define an Equation Goal that is a goal defined by an equation involving basic*

mathematical functions with existing goals and input data parameters 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.
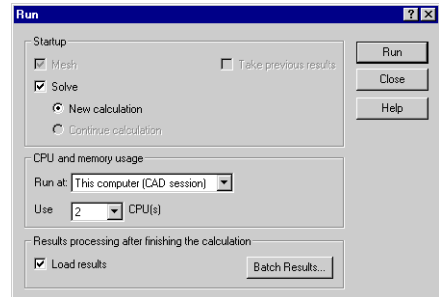
Click **File**, **Save**.

Solution

1 Click **Flow Simulation**, **Solve**, **Run**.

📖 *The already selected* **Load results** *check box means that the results will be automatically loaded after finishing the calculation.*
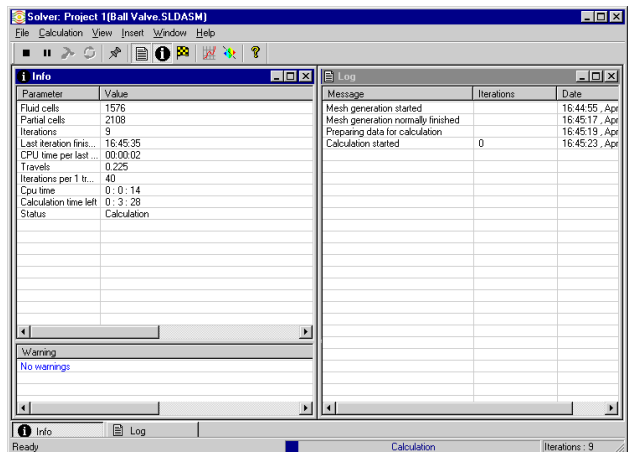
2 Click **Run**.

📖 *The solver takes less than a minute to run on a typical PC.*

Monitor the Solver

📖 *This is the solver monitor dialog box. By default, on the left is a log of each step taken in the solution process. On the right is the information dialog box with mesh information and warnings concerning the analysis. Do not be surprised when the error message "A vortex crosses the pressure opening" appears. We will explain this later during the demonstration.*

**1**   After the calculation has started and several first iterations has passed (keep your eye on the **Iterations** line in the **Info** window), click the **Suspend** ❚❚ button on the **Solver** toolbar.

 📖 *We employ the Suspend option only due to extreme simplicity of the current example, which otherwise could be calculated too fast, leaving you not enough time to perform the subsequent steps of monitoring. Normally you can use the monitoring tools without suspending the calculation.*

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

3   Select the **SG Average Static Pressure 1** in the **Select goals** list and click **OK**.

 📖 *This is the Goals dialog box and each goal created earlier is listed in the table at top. 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.*
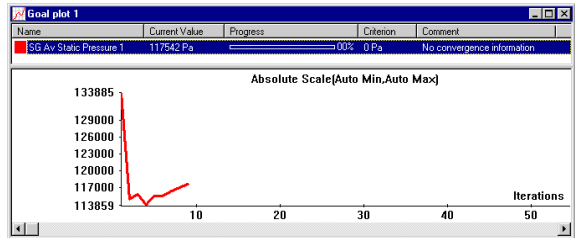
4   Click **Insert Preview** 🔀 on the **Solver** toolbar.

5   This is the **Preview Settings** dialog box. Selecting any SolidWorks plane from the **Plane name** list and pressing **OK** will create a preview plot of the solution in
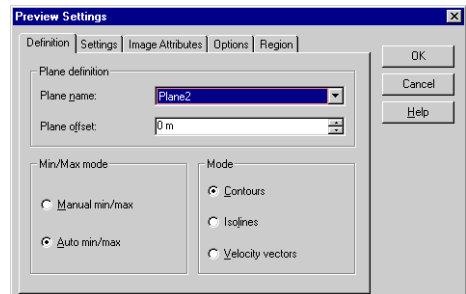
that plane. For this model `Plane2` is a good choice to use as the preview plane.

   📖 *The preview allows you 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.*
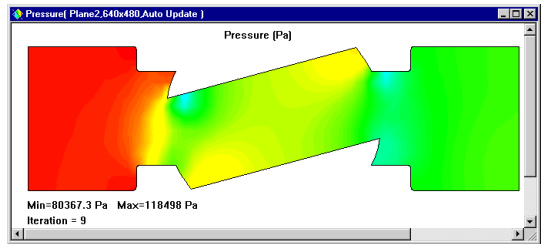
6 Click the **Suspend** ⏸ button again to let the solver go on.

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

Adjust Model Transparency

Click **Flow Simulation**, **Results**, **Display**, **Transparency**
and set the model transparency to 0.75.

   📖 *The first step for results processing is to create a
transparent view of the geometry, a 'glass-body'. This
way you can easily see where cut planes etc. are located
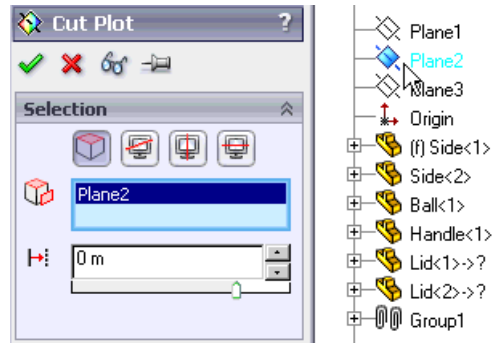with respect to the geometry.*
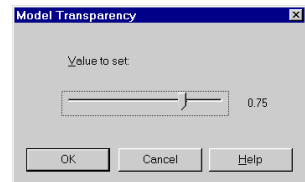
Cut Plots

1 In the Flow Simulation Analysis tree, right-click the **Cut
Plots** icon and select **Insert**.

2   Specify a plane. Choose `Plane 2` as the cut plane. To do this, in the flyout FeatureManager design tree select `Plane 2`.
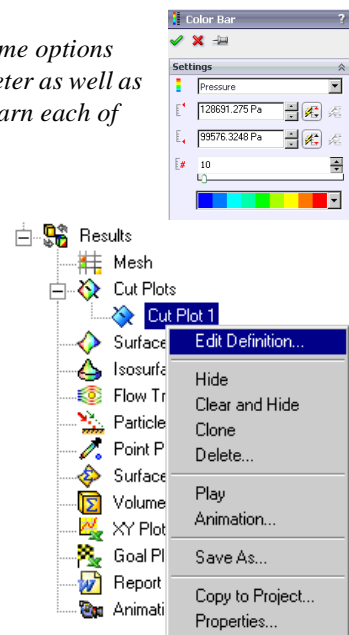
3   Click **OK** ✔ .

The plot you will see looks something like this.

📖 *A cut plot displays the distribution of the selected parameter on a certain SolidWorks plane. It can be represented as a contour plot, isolines, vectors, or as arbitrary combination of the above (e.g. contours with overlaid vectors).*
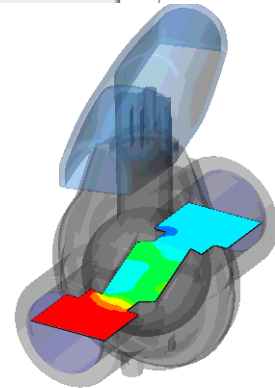
📖 *If you want to access additional options for this and other plots, you can double-click on the color bar. Some options available here include changing the displayed parameter as well as changing the min/max plot values. The best way to learn each of these options is thorough experimentation.*

4   Change the contour cut plot to a vector cut plot. To do this, right-click the **Cut Plot 1** icon and select **Edit Definition**.
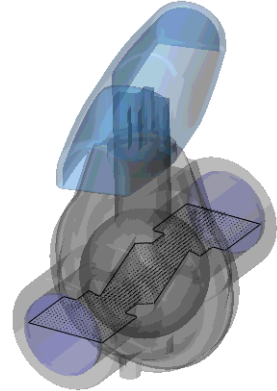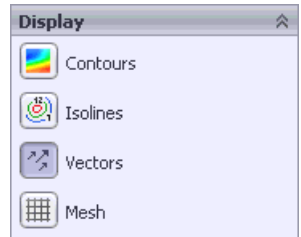
5 Under **Display**, clear **Contours** and select **Vectors** .

6 Click **OK** ✔.

This is the plot you should see.

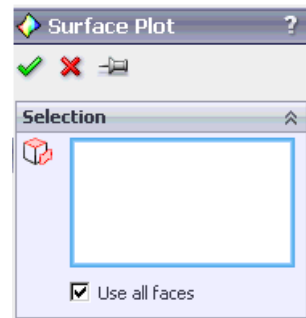   *The vectors size and spacing can be controlled under the **Vectors**.*

Surface Plots

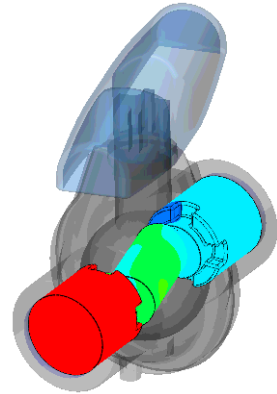Right-click the **Cut Plot 1** icon and select **Hide**.

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

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

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

3 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 **Insert**.

2 Keep the default value under **Value 1**.

3 Under **Appearance**, select **Grid** 🔴 and click **OK** ✔. This is the plot that will appear.

  📖 *The Isosurface is a 3-Dimensional surface created by Flow Simulation at a constant value for a specific variable.*

4 Right-click the **Isosurface 1** icon and select **Edit Definition**. Enable **Value 2** and specify some value in the appeared box that is different to the **Value 1**.

5 Click **OK** ✔ .
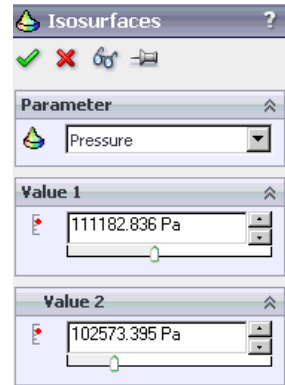
You should see something similar to this image.

📖 *The isosurface is a useful way of determining the exact 3D area, where the flow reaches a certain value of pressure, velocity or other parameter.*
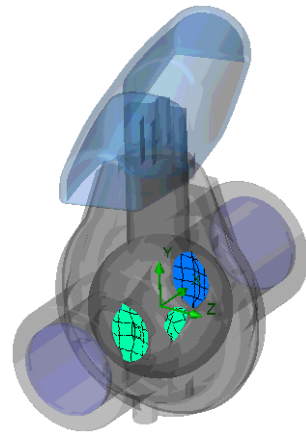
Flow Trajectory Plots
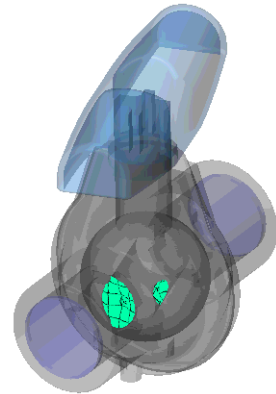
Right-click the **Isosurfaces** icon and select **Hide**.

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

2 Click the Flow Simulation Analysis Tree tab and
then click the **Static Pressure 1** item to select
the inner face of the **Lid <2>**.

3 Set the **Number of Points** 🔣 to 16.

4 Under **Appearance**, set **Draw Trajectories as**
🌊 **Bands**.
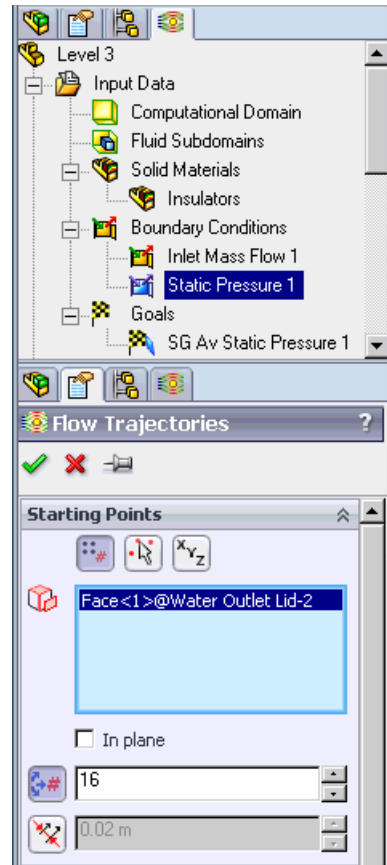
5 Click **OK** ✔ and your model should look like
the following:

📖 *Using Flow trajectories you can show the flow
streamlines. 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
Microsoft® Excel®. Additionally, you can save
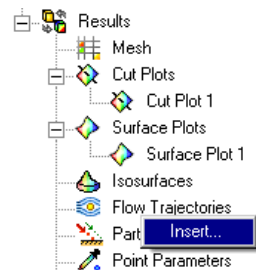trajectories as SolidWorks reference curves.*

📖 *For this plot we selected the outlet lid (any flat face or
sketch can be selected) and therefore every trajectory
crosses that selected face. Notice the trajectories that are
entering and exiting through the exit lid. This is the reason for the warning we received
during the calculation. Flow Simulation warns us of inappropriate analysis conditions
so that we do not need to be CFD experts. When flow both enters and exits the same*
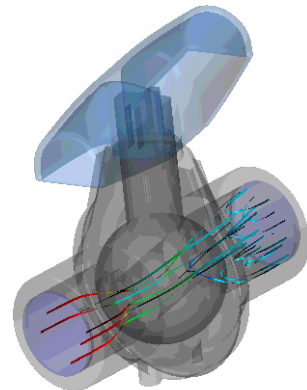
opening, the accuracy of the results will worsen. In a case like this, one would typically add the next component to the model (say, a pipe extending the computational domain) so that the vortex does not occur at opening.

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 calculation is finished. Take a look at Sketch1 in the FeatureManager design tree.
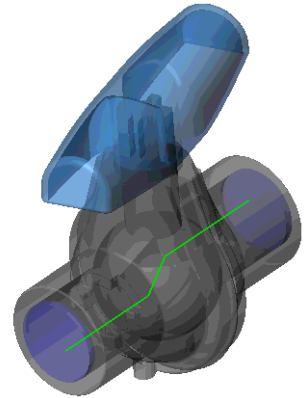
1 Right-click the **XY Plots** icon and select **Insert**.

2 Choose **Velocity** and **Pressure** as physical **Parameters**. Select **Sketch1** from the flyout FeatureManager design tree.

Leave all other options as defaults.
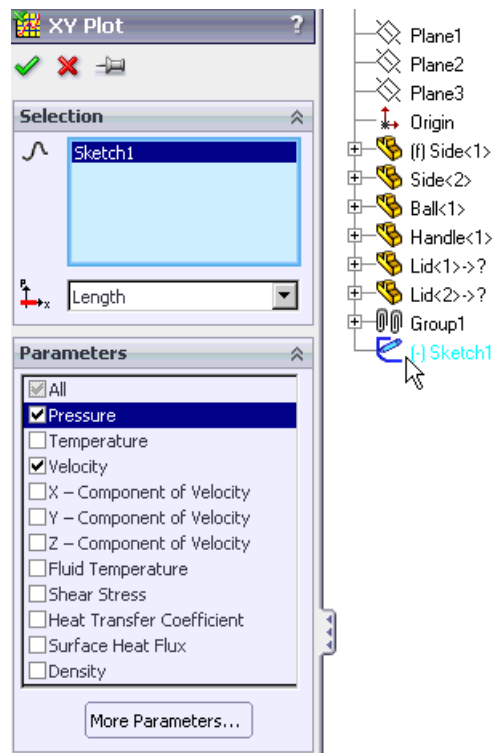
3 Click **OK** ✔. Excel will open and generate two columns of data points together with two charts for Velocity and for Pressure, respectively. One of these charts is shown below. You will need to toggle between different sheets in Excel to view each chart.

📖 *The XY Plot allows you to view any result along sketched lines. The data is put directly into Excel.*
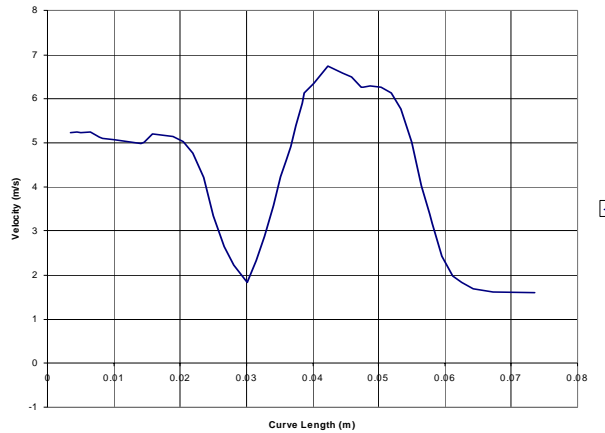
Surface Parameters

Surface Parameters is a feature used to determine the values of pressure, forces, heat fluxes as well as many other variables on any face in your model contacting the fluid. For this type of analysis, a calculation of the average static pressure drop from the valve inlet to outlet would probably be of some interest.

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

2 Click the Flow Simulation Analysis Tree tab and then click the **Inlet Mass Flow 1** item to select the inner face of the **Lid <1>**.

3 Under **Parameters**, select **All**.

4 Click **Show**. The calculated parameters values are displayed on the pane at the bottom of the screen. Local parameters are displayed at the left side of the bottom pane, while integral parameters are displayed at the right side.

5 Take a look at the local parameters.

| Local Parameter	Minimum	Maximum	Average	Bulk Average	Surface Area [m^2]
Pressure [Pa]	127597	127743	127670	127671	0.000391595
Density [kg/m^3]	997.562	997.562	997.562	997.562	0.000391595
Velocity [m/s]	1.61943	1.61943	1.61943	1.61943	0.000391595
X – Component of Velocity [m/s]	1.61943	1.61943	1.61943	1.61943	0.000391595
Y – Component of Velocity [m/s]	0	0	0	0	0.000391595
Z – Component of Velocity [m/s]	5.6542e-015	5.6542e-015	5.6542e-015	5.6542e-015	0.000391595
Fluid Temperature [K]	293.2	293.2	293.2	293.2	0.000391595

The average static pressure at the inlet face is shown to be about 127670 Pa. We already know that the outlet static pressure is 101325 Pa since we have specified it previously as a boundary condition. So, the average static pressure drop through the valve is about 27000 Pa.

**6** Close the **Surface Parameters** dialog.
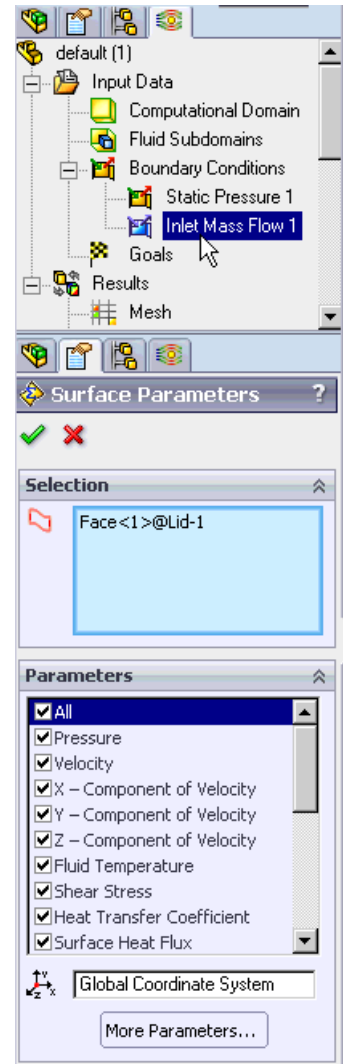
Analyze a Design Variant in the SolidWorks Ball part

    📖 *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 Flow Simulation and this 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.*
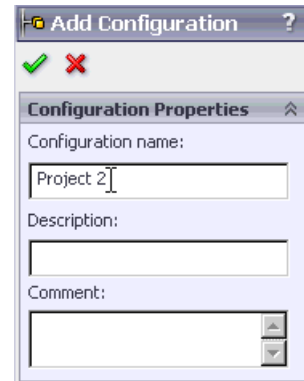
Create a new configuration using the SolidWorks Configuration Manager Tree.

1 Right-click the root item in the SolidWorks Configuration Manager and select **Add Configuration**.
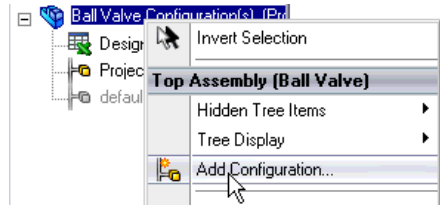
2 In the **Configuration Name** box type Project 2.

3 Click **OK** .

4 Go to FeatureManager design tree, right-click the **Ball** item and select **Open Part** 📂 . A new window **Ball.SLDPRT** appears.

Create a new configuration using the SolidWorks Configuration Manager Tree.
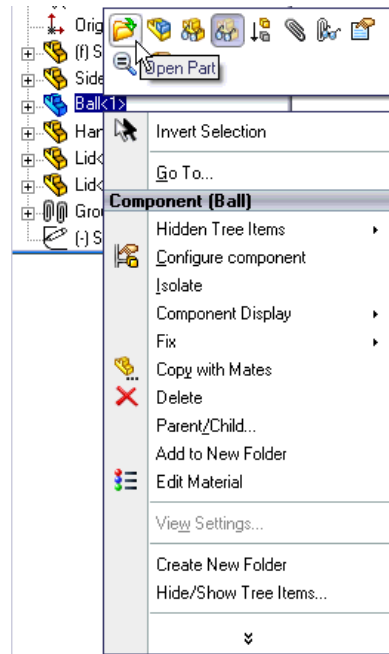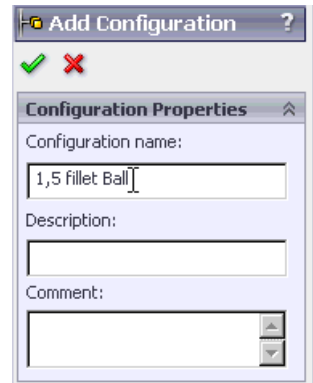
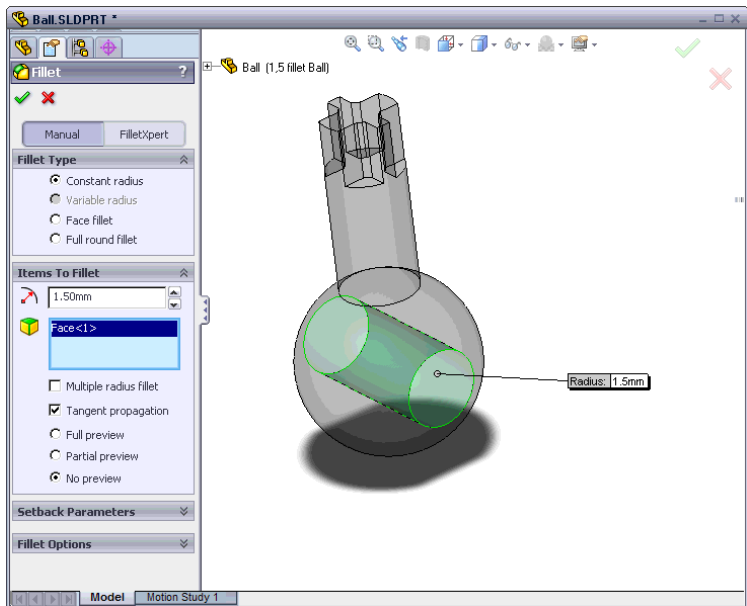**1** Right-click the root item in the SolidWorks Configuration Manager and select **Add Configuration.**
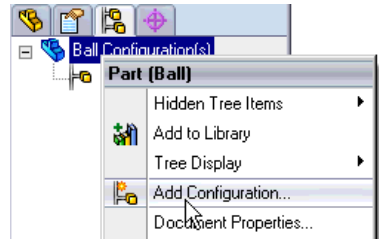
2 Name the new configuration as `1,5_fillet Ball`.

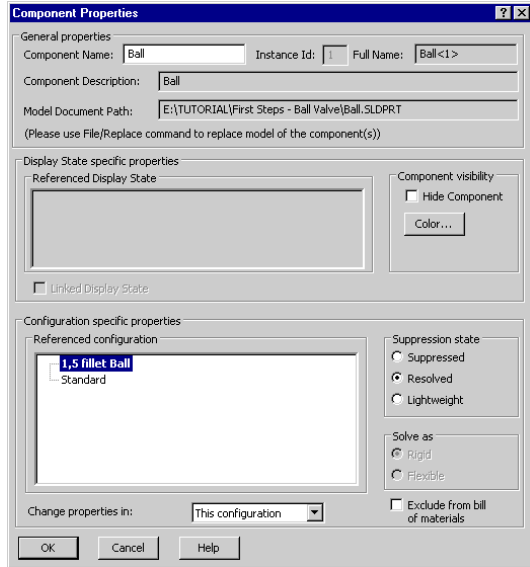**3** Click **OK** ✔ .

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

5   Switch back to the assembly
window and select **Yes** in the
message dialog box that appears. In
the FeatureManager design tree
right-click the **Ball** item and select

Component Properties.

6   At the bottom of the **Component
Properties** dialog box, under
Referenced configuration 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. Select **Yes**
for the message dialog box that appears.
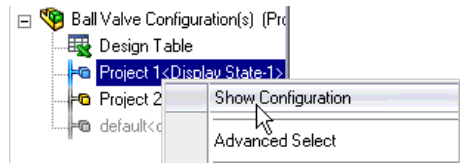
Clone the Project

1 Click **Flow Simulation**, **Project**, **Clone Project**.

2 Select **Add to existing**.

3 In the **Existing configuration** list select **Project 2**.
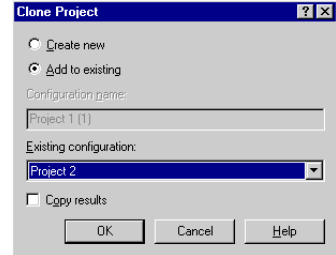
4 Click **OK**. Select **Yes** for each message dialog box that appears after you click **OK**.

   📖 *Now the Flow Simulation 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 Flow Simulation 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 **Flow Simulation**, **Project**, **Clone Project**.

2 Type Project 3 for the new project name and click **OK**.
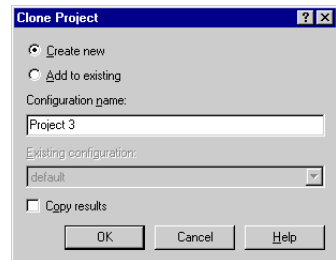
Flow Simulation now creates a new configuration. All our input data are copied, so we do not need to define our openings or goals again. The Boundary Conditions can be changed, deleted or added. All changes to the geometry will only be applied to this new configuration, so the old results remain valid. 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 outweighted the extra costs, how would you do this? Engineers have to make decisions such as this every day, and Flow Simulation is a tool to help them make those decisions. Every engineer who is required to make design decisions involving fluid and heat transfer should use Flow Simulation to test their ideas, allowing for fewer prototypes and quicker design cycles.