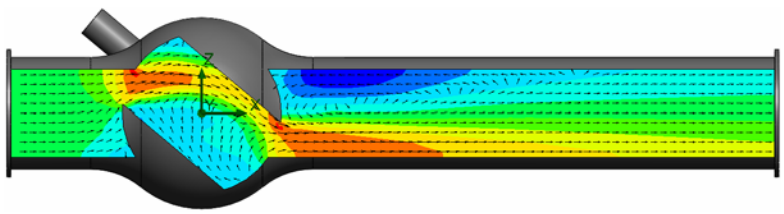
Engineering Design
and Technology Series

An Introduction to Flow Analysis Applications with SolidWorks Flow Simulation, Instructor Guide



Dassault Systèmes SolidWorks Corporation
300 Baker Avenue
Concord, Massachusetts 01742 USA
Phone: +1-800-693-9000

Outside the U.S.: +1-978-371-5011
Fax: +1-978-371-7303
Email: info@solidworks.com
Web: http://www.solidworks.com/education

Document Number: PME0418-ENG

i

Introduction

To the Instructor

This document introduces SolidWorks users to the SolidWorks Flow Simulation flow and heat transfer analyses software package. The specific goals of this lesson are to:

1. introduce the basic concepts of fluid flow analyses and their benefits
2. demonstrate the ease of use and the concise process for performing these analyses
3. introduce the basic rules for computational fluid dynamics analyses and how to obtain reliable and accurate results.

This document is structured similar to lessons in the SolidWorks Instructor Guide. This lesson has corresponding pages in the *SolidWorks Flow Simulation Student Workbook*.

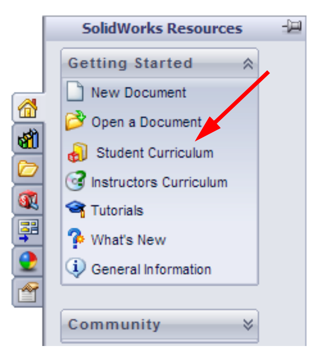
Note: This lesson does not attempt to teach all capabilities of SolidWorks Flow Simulation. It only intends to introduce the basic concepts and rules of performing flow and heat transfer analyses and to show the ease of use and the concise process of doing so.

Education Edition Curriculum and Courseware DVD

An *Education Edition Curriculum and Courseware* DVD is provided with this course.

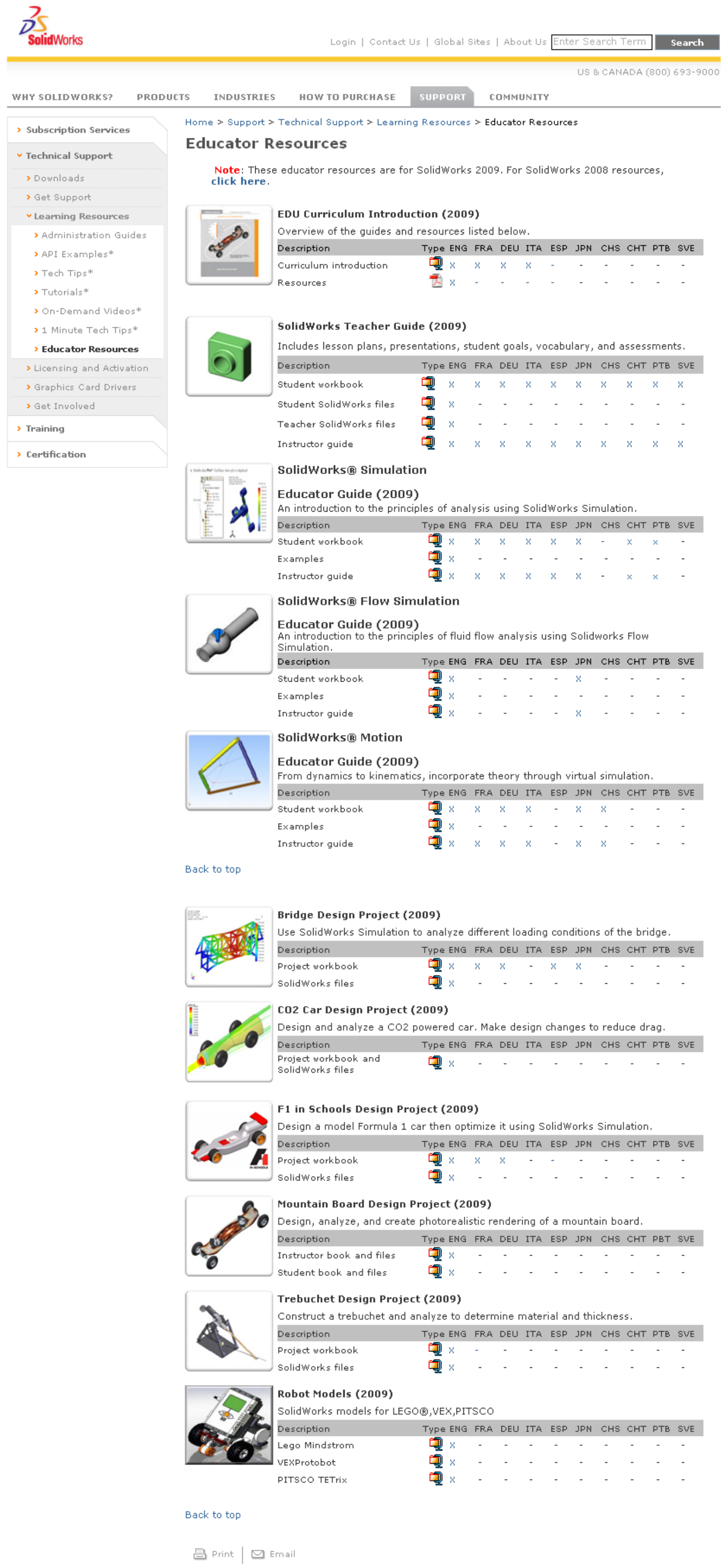
Installing the DVD creates a folder named `SolidWorks Curriculum_and_Courseware_2010`. This folder contains directories for this course and several others.

Course material for the students can also be downloaded from within SolidWorks. Click the SolidWorks Resources tab in the Task Pane and then select Student Curriculum.



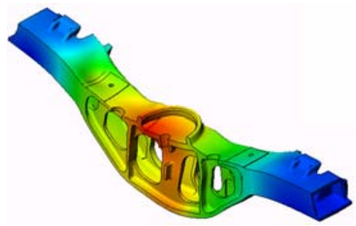
Double-click the course you would like to download. Control-select the course to download a ZIP file. The Lessons file contains the parts needed to complete the lessons. The Student Guide contains the PDF file of the course.

Course material for teachers can also be downloaded from the SolidWorks web site. Click the SolidWorks Resources tab in the Task Pane and then select Instructors Curriculum. This will take you to the Educator Resources page shown below.

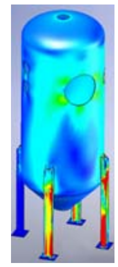


SolidWorks Simulation Product Line

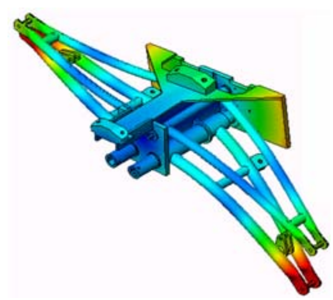
While this course focuses on the introduction to flow analysis using SolidWorks Flow Simulation, the full product line covers a wide range of analysis areas to consider. The paragraphs below lists the full offering of the SolidWorks Simulation packages and modules.



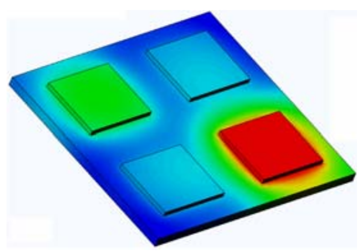
Static studies provide tools for the linear stress analysis of parts and assemblies loaded by static loads. Typical questions that will be answered using this study type are:
Will my part break under normal operating loads?
Is the model over-designed?
Can my design be modified to increase the safety factor?



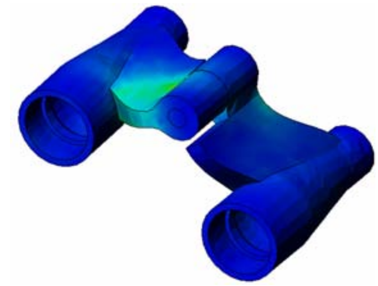
Buckling studies analyze performance of the thin parts loaded in compression. Typical questions that will be answered using this study type are:
Legs of my vessel are strong enough not to fail in yielding; but are they strong enough not to collapse due to loss of stability?
Can my design be modified to ensure stability of the thin components in my assembly?



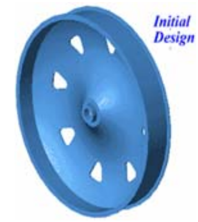
Frequency studies offer tools for the analysis of the natural modes and frequencies. This is essential in the design or many components loaded in both static and dynamic ways. Typical questions that will be answered using this study type are:
Will my part resonate under normal operating loads?
Are the frequency characteristics of my components suitable for the given application?
Can my design be modified to improve the frequency characteristics?



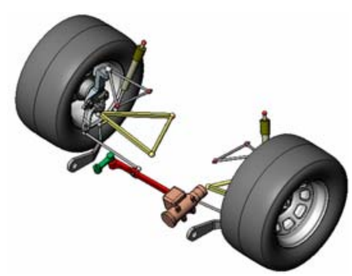
Thermal studies offer tools for the analysis of the heat transfer by means of conduction, convection, and radiation. Typical questions that will be answered using this study type are:
Will the temperatures changes effect my model?
How does my model operate in an environment with temperature fluctuation?
How long does it take for my model to cool down or overheat?
Does temperature change cause my model to expand?
Will the stresses caused by the temperature change cause my product failure (static studies, coupled with thermal studies would be used to answer this question)?



Drop test studies are used to analyze the stress of moving parts or assemblies impacting an obstacle. Typical questions that will be answered using this study type are:
What will happen if my product is mishandled during transportation or dropped?
How does my product behave when dropped on hard wood floor, carpet or concrete?

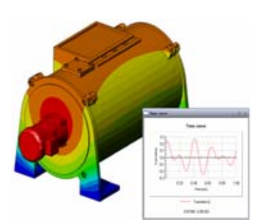


Optimization studies are applied to improve (optimize) your initial design based on a set of selected criteria such as maximum stress, weight, optimum frequency, etc. Typical questions that will be answered using this study type are:
Can the shape of my model be changed while maintaining the design intent?
Can my design be made lighter, smaller, cheaper without compromising strength of performance?



Fatigue studies analyze the resistance of parts and assemblies loaded repetitively over long periods of time. Typical questions that will be answered using this study type are:
Can the life span of my product be estimated accurately?
Will modifying my current design help extend the product life?
Is my model safe when exposed to fluctuating force or temperature loads over long periods of time?
Will redesigning my model help minimize damage caused by fluctuating forces or temperature?

Nonlinear studies provide tools for analyzing stress in parts and assemblies that experience severe loadings and/or large deformations. Typical questions that will be answered using this study type are:
Will parts made of rubber (o-rings for example) or foam perform well under given load?
Does my model experience excessive bending during normal operating conditions?



Dynamics studies analyze objects forced by loads that vary in time. Typical examples could be shock loads of components mounted in vehicles, turbines loaded by oscillatory forces, aircraft components loaded in random fashion, etc. Both linear (small structural deformations, basic material models) and nonlinear (large structural deformations, severe loadings and advanced materials) are available. Typical questions that will be answered using this study type are:
Are my mounts loaded by shock loading when vehicle hits a large pothole on the road designed safely? How much does it deform under such circumstances?

Motion Simulation enables user to analyze the kinematic and dynamic behavior of the mechanisns. Joint and inertial forces can subsequently be transferred into SolidWorks Simulation studies to continue with the stress analysis. Typical questions that will be answered using this modulus are:
What is the correct size of motor or actuator for my design?
Is the design of the linkages, gears or latch mechanisms optimal?
What are the displacemements, velocities and accelerations of the mechanism components?
Is the mechanism efficient? Can it be improved?

Composites modulus allows users to simulate structures manufactured from laminated composite materials.
Typical questions that will be answered using this modulus are:
Is the composite model failing under the given loading?
Can the structure be made lighter using composite materials while not compromising with the strength and safety?
Will my layered composite delaminate?

2

Basic Functionality of SolidWorks Flow Simulation

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 gas dynamic engineering problems. Upon successful completion of this lesson, the students should understand basic approaches to solving hydraulic and gas dynamic engineering problems. The students should see that the analysis of 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 SolidWorks Flow Simulation instead of conducting extremely time-consuming and expensive experimental works.
- Associate SolidWorks Flow Simulation 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 SolidWorks Flow Simulation's 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 SolidWorks Flow Simulation.
- Show the students the proper ways of correctly simulating real objects and flow phenomena with SolidWorks Flow Simulation.

  The results of analysis may vary slightly depending on versions/builds of SolidWorks and SolidWorks Flow Simulation.

Outline

- In Class Discussion
- Active Learning Exercise — Determination of Hydraulic Loss
  - Opening the `Valve.SLDPRT` Document
  - Checking the SolidWorks Flow Simulation menu
  - Model Description
  - Creating Lids Manually
  - Creating Lids Automatically
  - Creating a Project
  - SolidWorks Flow Simulation 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: Consumer products 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 SolidWorks Simulation) 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 SolidWorks Flow Simulation to perform fluid internal analysis on the `Valve.SLDPRT` part shown to the right.

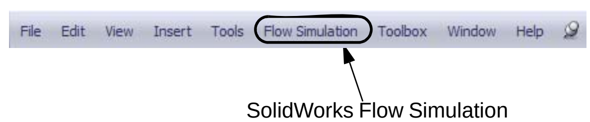
The step-by-step instructions are given below.

Opening the `Valve.SLDPRT` Document

1 Click **File**, **Open**. In the **Open** dialog box, browse to the `Valve.SLDPRT` part located in the corresponding subfolder of the `SolidWorks Curriculum_and_Courseware_2010` folder and click **Open** (or double-click the part).

Checking the SolidWorks Flow Simulation Menu

If SolidWorks Flow Simulation is properly installed, the Flow Simulation menu appears on the SolidWorks menu bar. If not:



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

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

2 Check the checkboxes next to SolidWorks Flow Simulation.

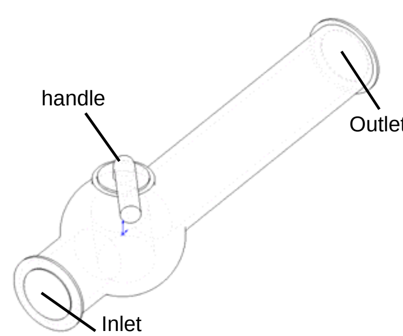
If SolidWorks Flow Simulation is not in the list, you need to install SolidWorks Flow Simulation first.

3 Click **OK**. The Flow Simulation 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^{o}. The Valve analysis represents a typical SolidWorks Flow Simulation internal analysis.

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

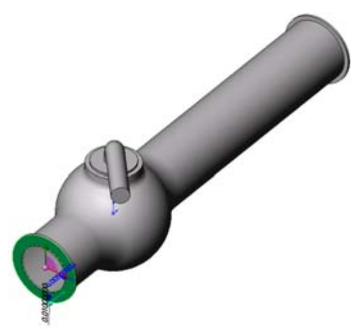
To perform an internal analysis all the model openings must be closed with lids, which are needed to specify inlet and outlet flow boundary conditions on them. In any case, the internal model space filled with a fluid must be fully closed. The lids are simply additional extrusions covering the openings. They can be created both manually and automacially; both of the procedures are shown below.

Creating Lids Manually

Creating Inlet Lid

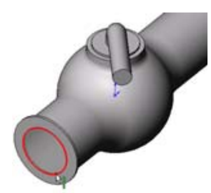
1 Select the face shown in the picture.

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



3 Select the tube's inner edge.

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



5 Complete the sketch by clicking **OK** button in the confirmation corner of the graphics area.

6 Click **Extruded Boss/Base** on the `Features` toolbar.

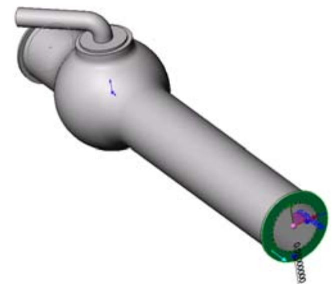
7 In the **Extrude** Feature PropertyManager change the settings as shown.

- **End Condition** = **Mid Plane**
- **Depth** = **0.005m**

8 Click to create the inlet lid.

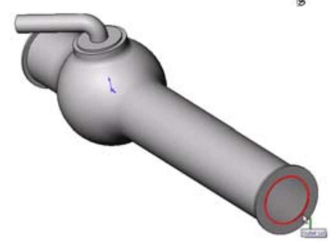
Next, in the same manner we will create the outlet lid.

Creating Outlet Lid



9 Select the face shown in the picture.

10 Click **Sketch** on the `Sketch` toolbar.



11 Select the tube's inner edge.

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

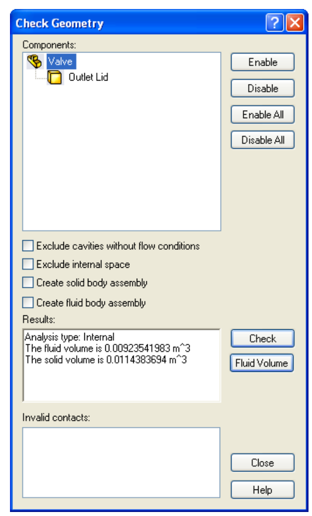
13 Rename the new extrusions `Extrude1` and `Extrude2` to `Inlet Lid` and `Outlet Lid`, correspondingly.

Not sure you have created the lids properly? SolidWorks Flow Simulation can easily check your model for possible geometry problems.

Checking the Geometry

1 To ensure the model is fully closed, click **Flow Simulation, 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 properly.



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.

Creating Lids Automatically

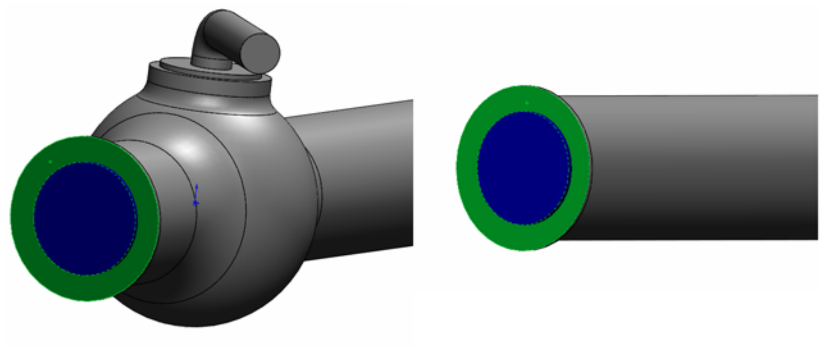
The previous step showed the manual lid creation. In the next step you will practice the SolidWorks Flow Simulation automatic lid creation tool. This tool can save considerable amount of time if multiple lids are needed to close the internal volume.

Deleting manually created lids

Delete `Inlet Lid` and `Outlet Lid` features.

Creating Inlet and Outlet Lids

1 Click **Flow Simulation**, **Tools**, **Create Lids**.
The **Create Lids** dialog box appears.



2 Select the two inlet and outlet faces shown in the figure.

3 Click  to complete the lid definitions.

4 Rename the newly created features `LID1` and `LID2` to `Inlet Lid` and `Outlet Lid`, respectively.

Note: In the assembly mode, each newly created lid forms a new part saved in the assembly folder.

The first step in performing flow analysis is to create a SolidWorks Flow Simulation project.

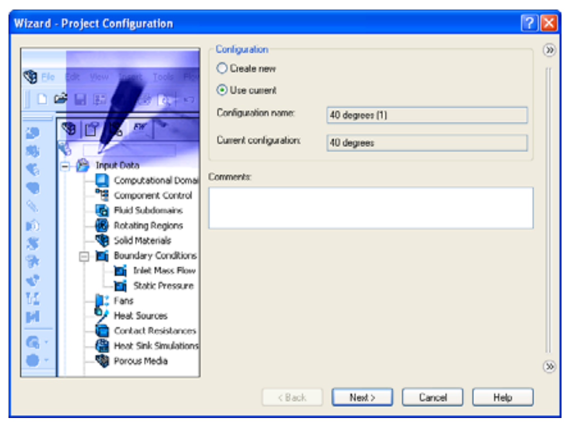
Creating a Project

1 Click **Flow Simulation, Project, Wizard**. The project wizard guides you through the definition of a new SolidWorks Flow Simulation project.

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

Each SolidWorks Flow Simulation 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.

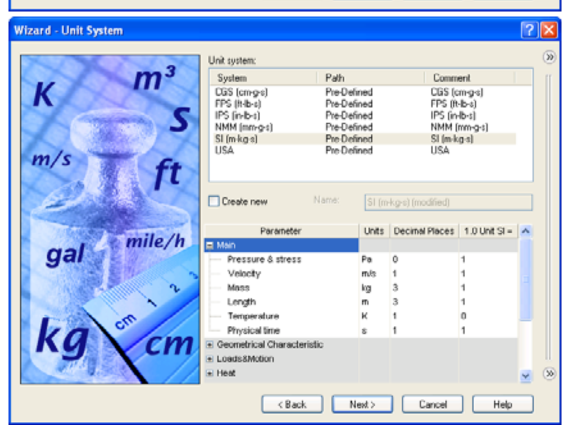
Click **Next**.



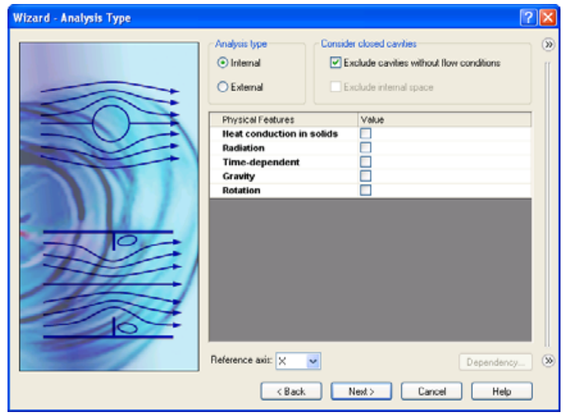
3 In the **Unit System** dialog box you can select the desired system of units for both input and output (results).

For this project we accept the default selection of **SI** (International System).

Click **Next**.

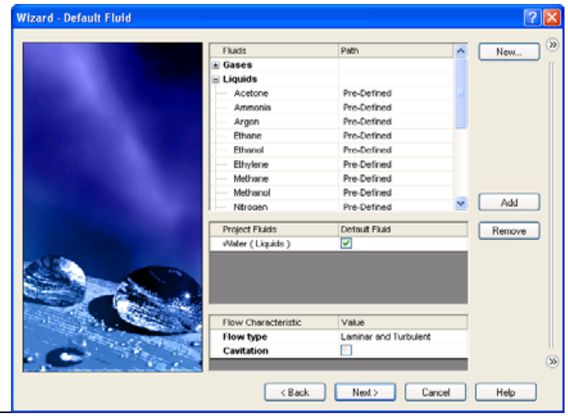


4 In the **Analysis Type** dialog box you can select either **Internal** or **External** type of the flow analysis. This dialog also allows you to specify advanced physical features you want to take into account: heat transfer in solids, surface-to-surface radiation, time-dependent effects, gravity and rotation.
Specify **Internal** type and accept the default values for the other settings. Click **Next**.



5 In the **Default Fluid** dialog box you can select the fluid type. The selected fluid type is assigned by default for all fluids in the analysis.

Click **Liquids** and then double-click the **Water** item in the **Liquids** list.
Leave defaults under **Flow Characteristics** and click **Next**.

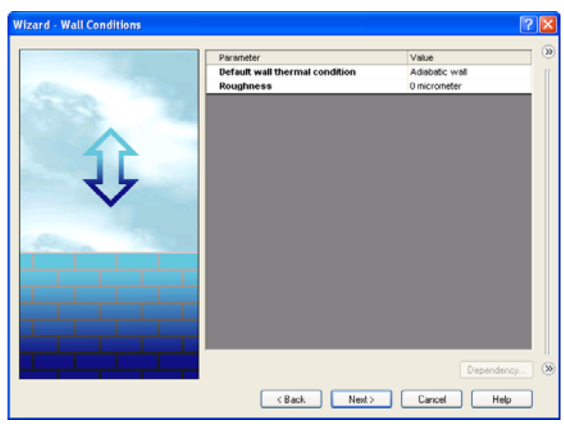


Note: The SolidWorks Flow Simulation **Engineering Database** contains physical properties of predefined and user-defined gases, real gases, incompressible liquids, non-Newtonian liquids, compressible liquids, solid substances and porous materials. It includes both constant values and tabular dependencies of various physical parameters on temperature and pressure.

The Engineering Database also contains unit systems, values of thermal contact resistance for various solid materials, properties of radiative surfaces and integral physical characteristics of some technical devices, namely, fans, heat sinks, and thermoelectric coolers. You can easily create your own substances, units, fan curves or specify a custom parameter you want to visualize.

6 In the **Wall Conditions** dialog box you can specify the wall roughness value and the wall thermal condition.

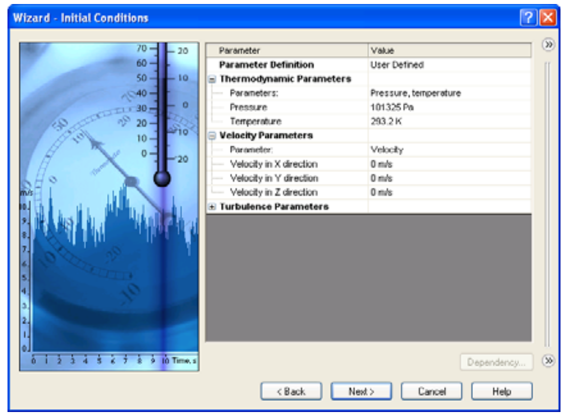
In this project we will not deal with the rough walls and heat conduction through the walls, so leave the default settings and click **Next**.



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

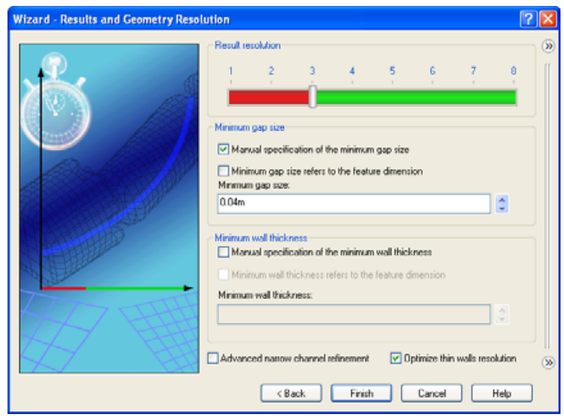
For this project use the default values.

Click **Next**.



Note: For steady flow problems SolidWorks Flow Simulation iterates until the solution converges. For unsteady (transient, or time-dependent) problems SolidWorks Flow Simulation marches in time for a period you specify.

8 In the **Results and Geometry Resolution** dialog box you can control the analysis accuracy as well as the mesh settings and, by this, the required computer resources (CPU time and memory).

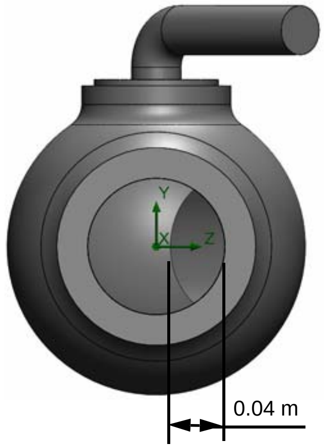


For this project accept the default **Result resolution** level **3**.

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: SolidWorks Flow Simulation 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.

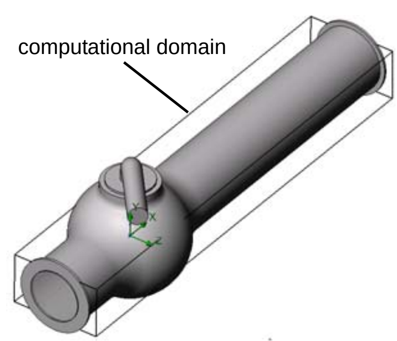
Click **Finish**.

SolidWorks Flow Simulation Design Tree

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

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

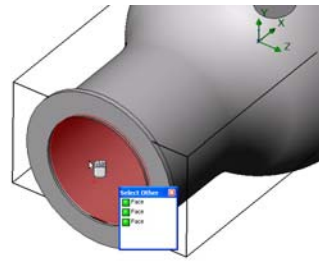
At the same time, in the SolidWorks graphics area a computational domain wireframe box appears. The flow and heat transfer calculations are performed inside the computational domain. The computational domain is a rectangular prism for both the 3D and 2D analyses. The computational domain boundaries are parallel to the global coordinate system planes.



Now let us specify the other parts of the project.

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

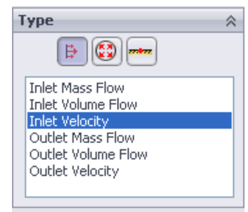
Specifying Boundary Conditions

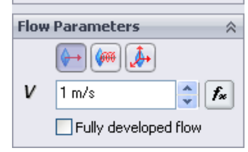


1 Click **Flow Simulation**, **Insert**, **Boundary Condition**.

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 the Boundary Condition** list.



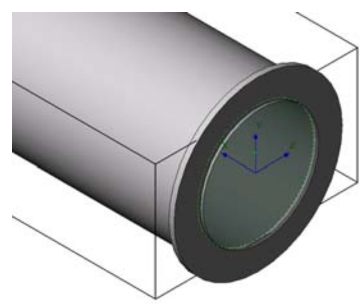


3 In the **Type** group box, click **Flow Openings** and select the **Inlet Velocity** item.

4 In the **Flow Parameters** group box, click **Normal to Face** item and set the **Velocity Normal to Face** *V* to **1 m/s** (just type the value, the units will appear automatically).

   Accept all other parameters and click .

By specifying this condition we define that the water enters the valve at the ball valve pipe inlet with the velocity of 1.0 m/s.



5 Select the `Outlet Lid` inner face.

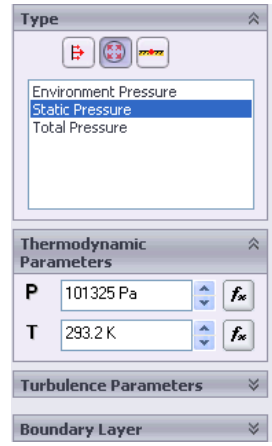
   In the graphics area, right-click outside the model and select **Insert Boundary Condition**. The **Boundary Condition** PropertyManager appears with the selected face in the **Faces to Apply the Boundary Condition** list.

Let us specify pressure on this boundary, otherwise the problem specification is deficient. Before the calculation starts, SolidWorks Flow Simulation 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 the problem solution.

6 Click **Pressure Openings** and in the **Type of Boundary Condition** list select the **Static Pressure** item.

7 Accept the default values for all of the other parameters (**101325 Pa** for **Static Pressure**, **293.2 K** for the **Temperature**, for example).

8 Click .



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

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

$$\xi = (dP) / \frac{\rho V^2}{2} = (dP) / P_{dyn}$$

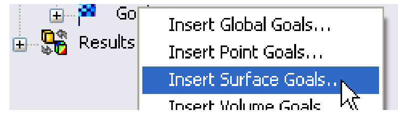
where ρ is water density, V is water inlet velocity, P_{dyn} is the dynamic pressure at inlet.

Since we already know the specified water velocity (1 $\frac{m}{s}$) and the water density (998.1934 $\frac{kg}{m^3}$ for the specified temperature of 293.2 K), our goal is to determine the total pressure value at the valve's inlet and outlet.

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

Engineering goals are the parameters which the user is interested in. Setting goals is essentially a way of conveying to SolidWorks Flow Simulation what you are trying to get out of the analysis, as well as means of reducing the time SolidWorks Flow Simulation takes to reach a solution. By only selecting the variable which the user desires accurate values for, SolidWorks Flow Simulation 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 defined over the entire domain (Global Goals), within a selected volume (Volume Goal), on a selected area (Surface Goal) or at a specific point of the model (Point Goal). Furthermore, SolidWorks Flow Simulation can consider either average, minimum or maximum parameter value to define the goal. You can also define an Equation Goal that is a goal defined by an equation (involving 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.

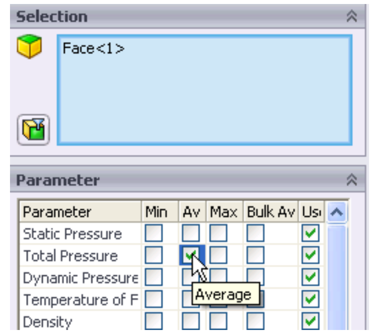
Specifying Surface Goals



1 In the SolidWorks Flow Simulation 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 Velocity 1` item in the SolidWorks Flow Simulation design tree. The face related to the specified boundary condition is automatically selected and appears in the **Faces to Apply the Surface Goal** list.

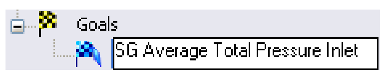


3 In the **Parameter** list, find **Total Pressure**. Click in the **Av** column to use the average value and keep selected **Use for conv.** to use this goal for the convergence control.

Note: To see the parameter names more clearly, you will probably find useful to enlarge the PropertyManager area by dragging the vertical bar to the right.

4 Click .

5 In the SolidWorks Flow Simulation design tree click-pause-click the new `SG Av Total Pressure 1` 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**.

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

7 Click the `Static Pressure 1` item in the SolidWorks Flow Simulation design tree to select the inner face of the `Outlet Lid`.

8 In the **Parameter** list, find **Total Pressure**.

9 Click in the **Av** column and then click .

10 Click-pause-click the new `SG Av Total Pressure 1` item and rename it to `SG Average Total Pressure Outlet`.

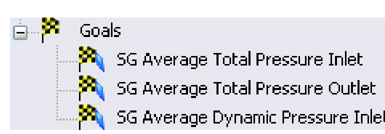
11 Right-click the `Goals` icon again and select **Insert Surface Goal**.

12 Click the `Inlet Velocity 1` item to select the inner face of the `Inlet Lid`**.**

13 In the **Parameter** list, find **Dynamic Pressure**.

14 Click in the **Av** column and then click .

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



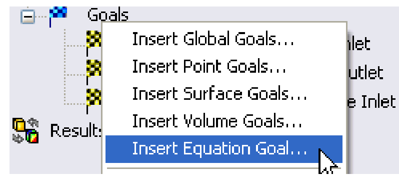
The value of the dynamic pressure at the inlet can be calculated manually. We have specified the dynamic pressure goal just for the convenience of the further calculation of hydraulic losses.

After finishing the calculation you will need to manually calculate the hydraulic loss ξ from the obtained total pressures values. Instead, let SolidWorks Flow Simulation 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 monitored during the calculation and while displaying results in the same way as the other goals. Any of the existing goals can be used as variables, including other equation goals, except those 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.

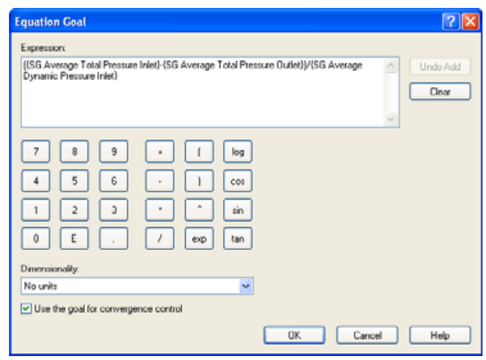


2 Click the left bracket button (or type "**(**".

3 In the `Goals` list select the `SG Average Total Pressure Inlet` goal. The goal is then automatically added in the **Expression** field.

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

5 In the `Goals` list select the `SG Average Total Pressure Outlet` goal.



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

7 In the `Goals` list select 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 existing 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. SolidWorks Flow Simulation has no information about the physical meaning of the specified constants so you need to specify the displayed dimensionality manually.

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

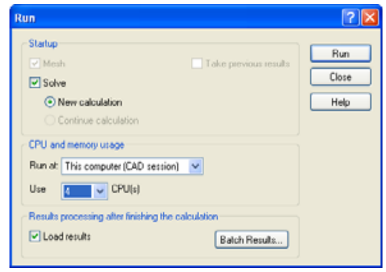
10 Rename it to `Hydraulic Loss`.

Now the SolidWorks Flow Simulation project is ready for the calculation. SolidWorks Flow Simulation 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

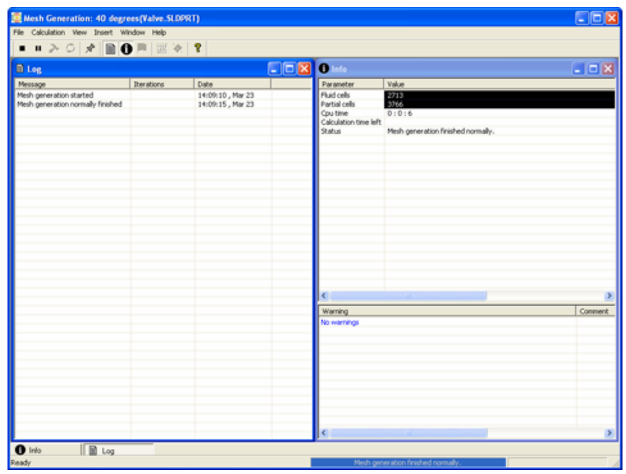
1 Click **Flow Simulation**, **Solve**, **Run**. The **Run** dialog box appears.

2 Click **Run** to start the calculation.



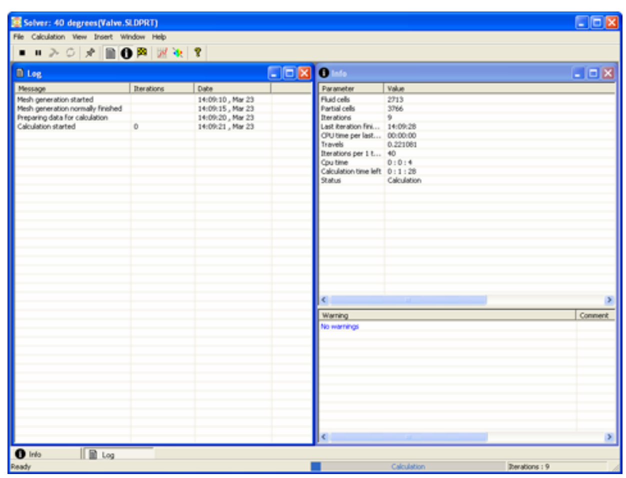
The calculation should take about 2 minutes to run on a 2.26 GHz Pentium M computer.

SolidWorks Flow Simulation 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 cells, i.e. elementary rectangular volumes. The cells are further subdivided as necessary to resolve properly 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. To the left you may see the stepwise log of the solution process. The information dialog box arranged to the right contains summary information on the mesh and any warnings on different issues that may arise during the analysis.

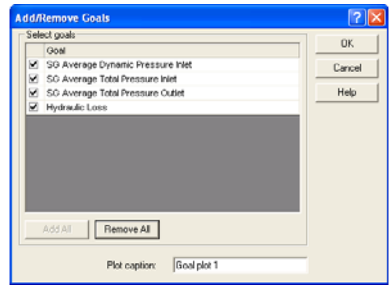


During the calculation you can monitor the convergence behavior of your goals (Goal Plot), view the current results in the specified plane (Preview) and display the minimum and maximum parameter values at the current iteration (Min/Max table).

Creating Goal Plot

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

2 Click **Add All** to check all goals and click **OK**.



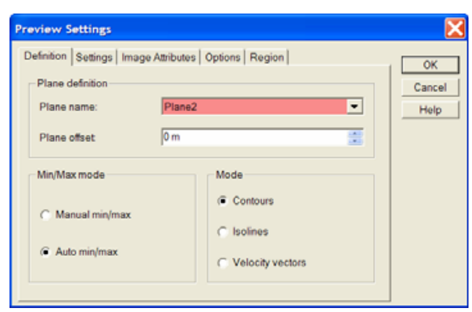
This is the Goal Plot dialog box. All added goals together with their current values are listed at the top part of the window, as well as the current progress towards completion given as a percentage. The progress value is only an estimate and generally (but not necessarily) increases with time. Below you can see the graph of all goals.

Convergence is an iterative process. The discretization of the flow field imposes conditions on each parameter and each parameter cannot reach an absolutely stable value but will oscillate near this value from iteration to iteration. When SolidWorks Flow Simulation analyzes the goal's convergence, it calculates the goal's dispersion defined as the difference between the goal's maximum and minimum values over the analysis interval reckoned from the last iteration and compares this dispersion with the goal's convergence criterion dispersion, either specified by you or automatically determined by SolidWorks Flow Simulation. Once the oscillations are less than the convergence criterion the goal becomes converged.

Preview Results

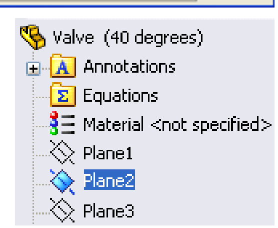
1 While the calculation is still running, click **Insert Preview** on the `Solver` toolbar. The **Preview Settings** dialog box appears.

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 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 contours, isolines 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

Expand the `Results` folder in the project tree by clicking the corresponding (+) sign.

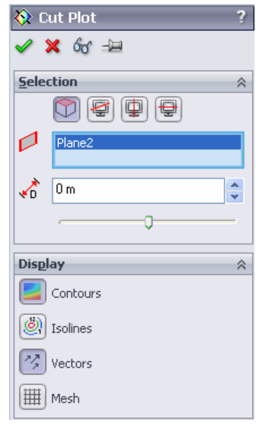
Note: When the solver is finished, the results are loaded automatically (unless the **Load results** check box in the **Run** window has been unchecked). However, when working with a previously calculated project, you need to load the results manually by clicking **Flow Simulation**, **Results**, **Load/Unload Results**.

Once the calculation finishes, you can view the saved calculation results in numerous ways and in a customized manner directly within the graphics area. The `Result` folder features functions that may be used to view your results: `Cut Plots` (section views of parameter distribution), `3D-Profile Plots` (section views in relief representation), `Surface Plots` (distribution of a parameter on a selected surface), `Isosurfaces`, `Flow Trajectories`, `Particle Studies` (particle trajectories), `XY Plots` (diagrams of parameter behavior along a curve or sketch), `Point Parameters` (getting parameters at specified points), `Surface Parameters` (getting parameters at specified surfaces), `Volume Parameters` (getting parameters within specified volumes), `Goals` (behavior of the specified goals during the calculation), `Reports` (export of 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.

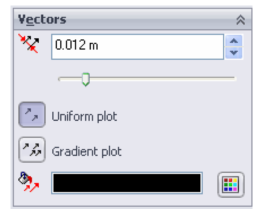
The Cut Plot displays results of a selected parameter in a selected view section. To define the view section, you can use SolidWorks planes or model planar faces (with the additional shift if necessary). The parameter values can be represented as a contour plot, isolines, vectors, or in a combination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



2 Click the SolidWorks FeatureManager and select `Plane2`. Its name appears in the **Section Plane or Planar Face** list on the **Selection** tab.

3 In the **Cut Plot** PropertyManager window, in addition to displaying **Contours** , click **Vectors** .

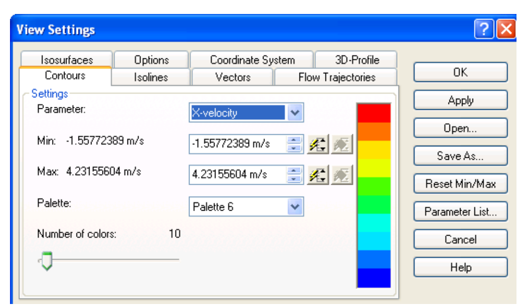
4 In the **Vectors** group box, using the slider set the **Vector Spacing** to approximately **0.012 m**.



5 Click **View Settings** in order to specify the parameter which will be shown in 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-Component of Velocity**.



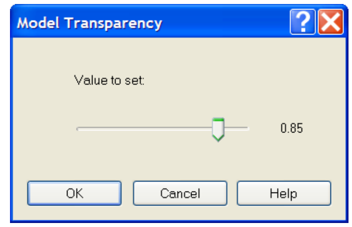
7 Click **OK** to save changes and close the **View Settings** dialog box.

8 Click to create the cut plot. The new `Cut Plot 1` item appears in the SolidWorks Flow Simulation 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 **Flow Simulation**, **Results**, **Display**, **Geometry** (alternatively, you can use the standard SolidWorks **Section View** option) or change the model transparency (as is done in the next step below).

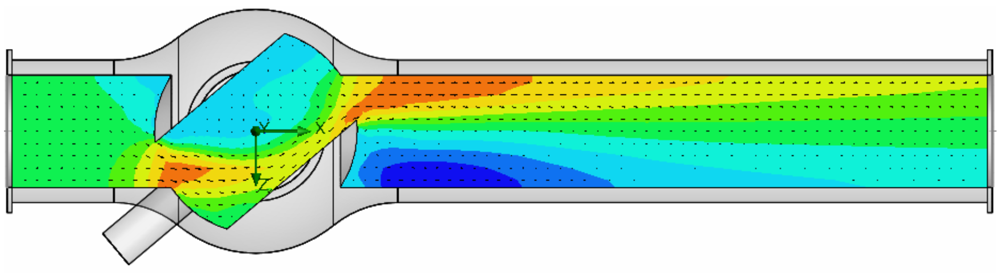
9 Click the **Flow Simulation**, **Results**, **Display**, **Geometry** to show the model. Click **Flow Simulation**, **Results**, **Display**, **Transparency** and drag the slider to set the value of approximately **0.85**.

Click ✔.



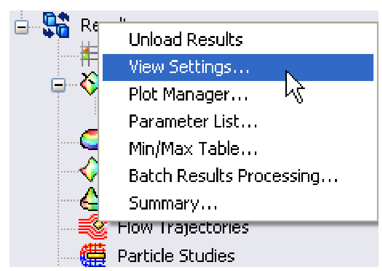
10 In the SolidWorks Flow Simulation design tree, right-click the `Computational Domain` icon and select **Hide**.

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:

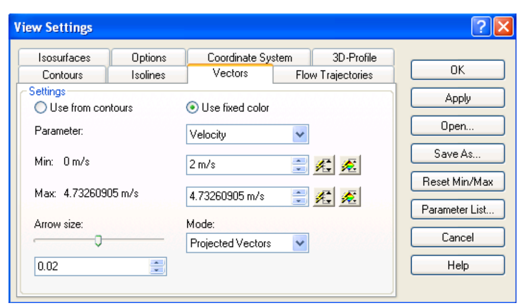
11 In the SolidWorks Flow Simulation design tree, right-click the `Results` icon and select **View Settings**.



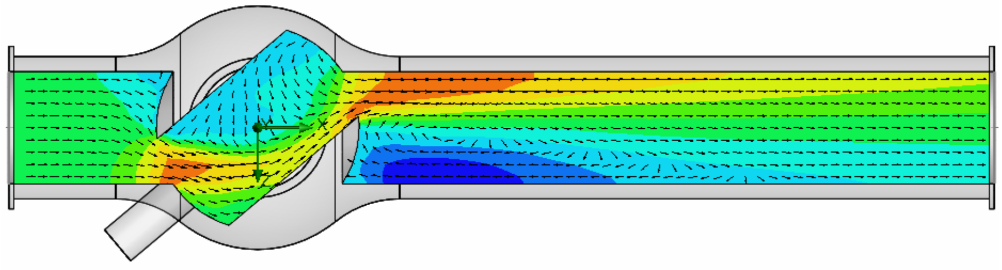
12 In the **View Settings** dialog, click the **Vectors** tab and type **0.02 m** in the **Arrow size** box.

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.

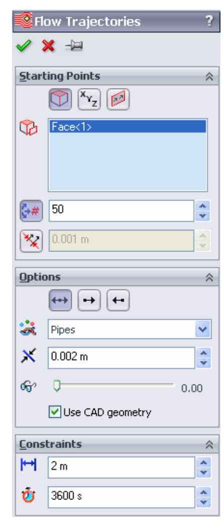


Displaying Flow Trajectories

With the use of **Flow trajectories** you can show the flow streamlines. Flow streamlines provide a very clear and comprehensible representation of the flow peculiarities. 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.



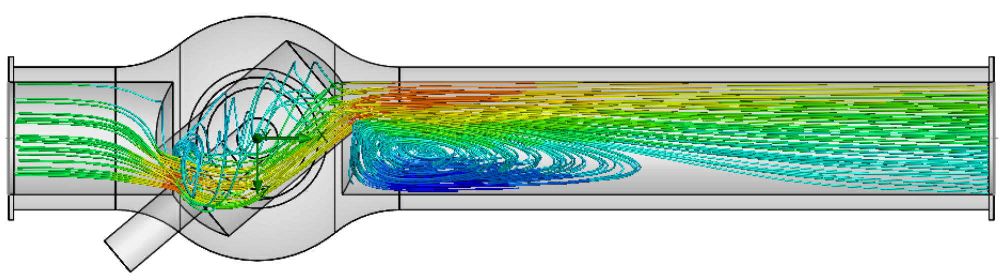
2 In the SolidWorks Flow Simulation Design Tree, click the `Static Pressure 1` item to select the inner face of the `Outlet Lid`. Trajectories launched from the outlet opening will better visualize the vortex occurring downstream the valve's obstacle.

3 Set the **Number of trajectories** to **50**.

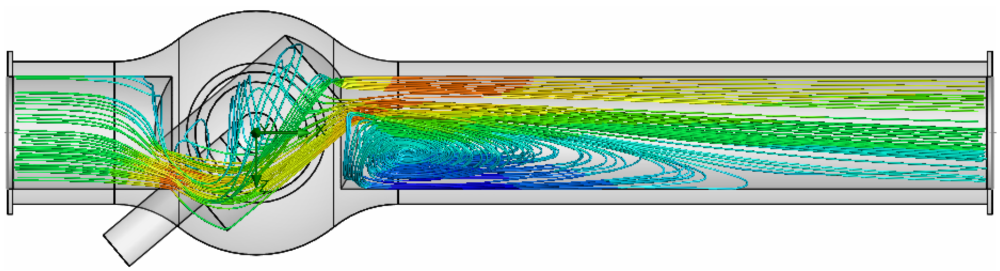
4 Click the **Constraints** tab and decrease the **Maximum length** of trajectories to **2 m**.

5 Click **OK** to display trajectories.

6



Some may prefer to display the flow trajectories with the help of a section plot. Use `Plane2` to define a section plot to show the flow trajectories.



Rotate the model to examine the 3D structure of the vortices in more detail.

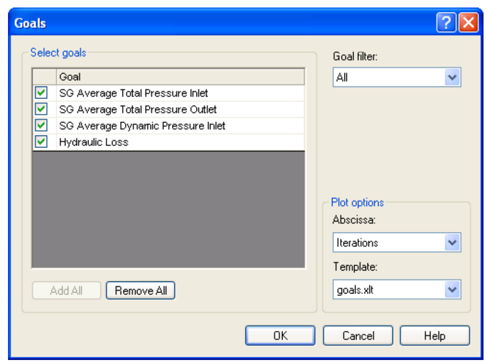
Creating a Goal Plot

The Goal Plot allows you to study the goal changes in the course of the calculation. SolidWorks Flow Simulation uses Microsoft Excel to display the goal plot data. Each goal plot is displayed in a separate sheet. The converged values of all project goals are displayed in the `Summary` sheet of an automatically created Excel workbook.

1 In the SolidWorks Flow Simulation design tree, under `Results`, right-click the `Goals` icon and select **Insert**. The **Goals** dialog box appears.

2 Click **Add All**.

3 Click **OK**. The `goals1` Excel workbook is created.



This workbook displays how the goal values had changed during the calculation. You can take the total pressure value presented in the `Summary` sheet.

Valve.SLDPRT [40 degrees]

Goal Name	Unit	Value	Averaged Value	Minimum Value	Maximum Value	Progress [%]	Use In Convergence	Delta	Criteria
SG Average Total Pressure	[Pa]	112270.2791	112275.2008	112268.6415	112294.8531	100	Yes	26.21163269	543.7731616
SG Average Total Pressure	[Pa]	101968.6227	101976.6097	101968.6227	101996.0177	100	Yes	27.39501123	29.72826268
SG Average Dynamic Pres	[Pa]	498.7808697	498.7808697	498.7808697	498.7808697	100	Yes	0	4.98781E-06
Hydraulic loss	[ ]	20.65367186	20.64752624	20.62365726	20.66256681	100	Yes	0.03890955	1.076820996

Cloning Project

The current calculation yields the total hydraulic resistance ξ including both valve's hydraulic resistance $\xi\nu$ (due to the obstacle) and the tubes' hydraulic resistance due to friction ξ_f : $\xi = \xi\nu + \xi_f$. To obtain the valve's resistance, it is necessary to subtract from the obtained data the total pressure loss due to friction in a straight pipe of the same length and diameter. To do that, we will perform the same calculations in the ball valve model whose handle is turned by an angle of 0^o.

You can create a new SolidWorks Flow Simulation project in three ways:

- The Project Wizard is the most straightforward way of creating a SolidWorks Flow Simulation project. It guides you step-by-step through the analysis set-up process.
- To analyze different flow or model variations, the most efficient method is to clone (copy) your current project. The new project will have all the settings of the cloned project, optionally including the results settings.
- You can create a SolidWorks Flow Simulation project by using a Template, either a default template or custom template created from a previous SolidWorks Flow Simulation project. Template contains only general project settings (the settings you specify in the Wizard and General Settings only) and does not contain the other project features like boundary conditions, goals, etc.

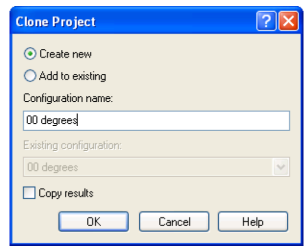
The easiest way to create a new SolidWorks configuration for 0^{o} angle and specify the same condition as the 40^{o} angle project is to clone the existing **40 project**.

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

2 Click **Create New**.

3 In the **Configuration name** box, type `00 degrees`.

4 Click **OK**.



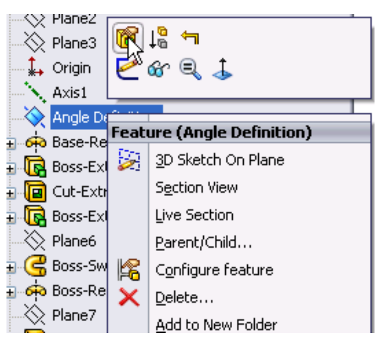
Now the new SolidWorks Flow SimulationSolidWorks Flow Simulation project is attached to the new `00 degrees` configuration and has inherited all the settings from the `40 degrees` project. All our input data are copied, so we do not need to redefine them. All changes will only be applied to this new configuration, not affecting the old project and its results.

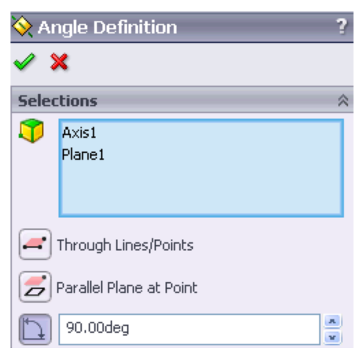
Changing the Valve Angle

1 In the SolidWorks FeatureManager, right-click the `Angle Definition` feature and select **Edit Feature**.

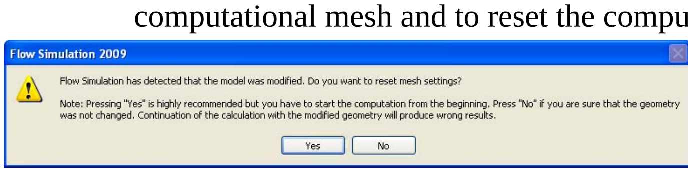
2 In the **At angle** box, type **90**.

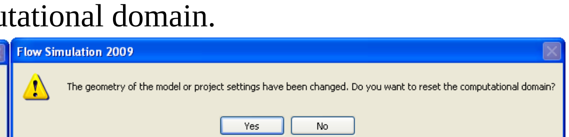
3 Click **OK** .





After clicking **OK**, two warning messages appear asking you to rebuild the computational mesh and to reset the computational domain.





4 Answer **Yes** to the both messages.

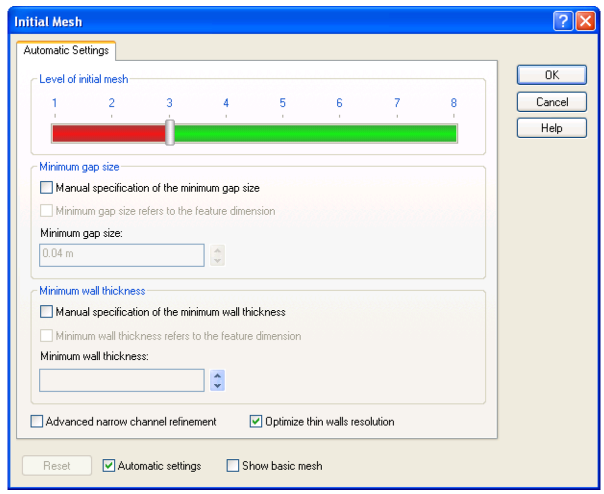
Changing the Geometry Resolution

Since at the zero angle the ball valve becomes a simple straight pipe, there is no need to set the **Minimum gap size** value smaller than the default gap size which, in our case, is automatically set equal to the pipe's diameter (the automatic minimum gap size depends on the characteristic size of the faces on which the boundary conditions are set). Note that using a smaller gap size will result in a finer mesh which, in turn, will require more CPU time and memory. To solve your task in the most effective way you should choose the optimal settings for the task.

1 Click **Flow Simulation**, **Initial Mesh**.

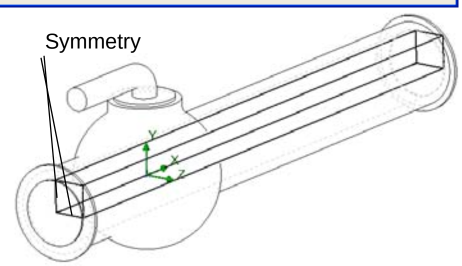
2 Clear the **Manual specification of the minimum gap size** check box.

3 Click **OK**.



Changing the Computational Domain

You can take advantage of the symmetry of the straight pipe to reduce the CPU time and memory requirements for the computation. Since the flow is symmetric at two directions (Y and Z), it is possible to "cut" the model in one fourth and use a symmetry boundary condition on the planes of symmetry. This procedure is not required but is recommended for efficient analyses.



Note: The symmetric conditions can be applied only if you are sure that the flow is symmetric. Note that sometimes symmetry of both the model and the incoming flow does not guarantee symmetry in other flow regions, e.g. a von Karman vortex street behind a cylinder. In our case, the flow in the straight pipe is symmetric so we can reduce the computational domain.

1 In the SolidWorks Flow Simulation design tree right-click the `Computational Domain` icon and select **Edit Definition**. The **Computational Domain** dialog box appears.

In the **Computational Domain** dialog box you can perform the following:

- Resize the **Computational Domain**.
- Apply the **Symmetry** boundary condition. The flow symmetry planes can be utilized as computational domain boundaries with specified **Symmetry** conditions on them.

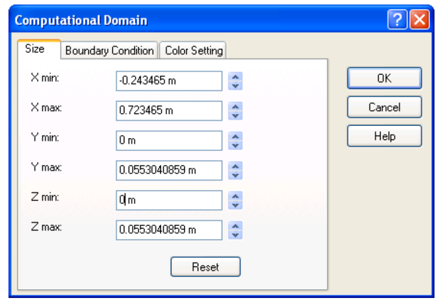
In this case, the computational domain boundaries must coincide with the flow symmetry planes.

- Specify a **2D plane flow**. If you are fully confident that the flow is a 2D plane flow, you can redefine the computational domain from the default 3D analysis to a 2D plane flow analysis that results in decreases in memory and CPU time requirements. To activate a 2D planar analysis, select **2D plane flow** on the **Boundary Condition** tab.

2 In the **Y min** box type **0**.

3 In the **Z min** box type **0**.

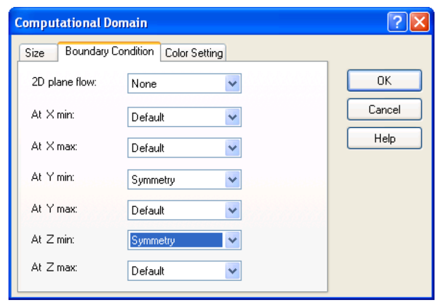
4 Click the **Boundary Condition** tab.



5 In the **At Y min** and **At Z min** lists select **Symmetry**.

6 Click **OK**.

7 Click **Flow Simulation**, **Solve**, **Run**. Then click **Run** to start the calculation.



Getting the Valve's Hydraulic Loss

After the calculation is finished, close the monitor dialog box and create the goal plot with the newly obtained results.

Valve.SLDPRT [00 degrees]

	Unit	Value	Averaged Value	Minimum Value	Maximum Value	Progress [%]	Use In Convergence	Delta	Criteria
SG Average Total Pressure	[Pa]	101905.9009	101913.4966	101905.9009	101940.1773	100	Yes	34.27637043	149.6592868
SG Average Total Pressure	[Pa]	101811.8221	101811.9443	101810.101	101812.656	100	Yes	2.554935328	2.91454975
SG Average Dynamic Pres	[Pa]	498.7808697	498.7808697	498.7808697	498.7808697	100	Yes	0	4.98781E-06
Hydraulic loss	[ ]	0.188617473	0.203601043	0.188617473	0.26078832	100	Yes	0.072170847	0.299347039

Now you can calculate the valve's hydraulic loss in the ball valve whose handle is turned by 40^{o}. To determine the parameter's steady-state value more accurately, it would be more accurate to use the values averaged over the analysis interval, which are shown in the `Averaged Value` column.

Total hydraulic losses (40 deg)	Friction losses (0 deg)	Valve's loss
20.6	0.20	20.4

Save Your Work and Exit SolidWorks

1 Click on the Standard toolbar or click **File**, **Save**.

2 Click **File**, **Exit** on the Main menu.

5 Minute Assessment – Answer Key

1 What is SolidWorks Flow Simulation?

Answer: SolidWorks Flow Simulation is a fluid flow and heat transfer analysis product fully integrated within SolidWorks.

2 How do you start a SolidWorks Flow Simulation session?

Answer: On the Windows task bar, click **Start**, **Programs**, **SolidWorks**, **SolidWorks Application**. The SolidWorks application starts.

3 What is a fluid flow analysis?

Answer: Fluid flow analysis is a process to simulate how the fluid influences the design and performance of your device or to simulate how a device affects the fluid flow parameters.

4 Why analysis is important?

Answer: Analyses enables you to understand and improve your design saving your time and money by reducing traditional design cycle.

5 What kind of analyses is typical for SolidWorks Flow Simulation internal flow analyses?

Answer: Typical internal analyses are those when the fluid enters a model at the model inlets and exits the model through outlets.

6 What is the specific requirement of SolidWorks Flow Simulation internal analyses?

Answer: SolidWorks Flow Simulation internal analyses require that the models be fully closed.

7 How can you ensure the model is closed?

Answer: You can calculate the model’s internal volume by using the **Check Geometry** tool. If the volume is zero, your model is not closed.

8 Why is it necessary to add lids to the ball valve model openings?

Answer: Lids make the model closed for an internal analysis. You will need to apply inlet and outlet boundary conditions on the lids.

9 What is the first step to start a SolidWorks Flow Simulation analysis?

Answer: The first step to start a SolidWorks Flow Simulation analysis is to create a SolidWorks Flow Simulation project.

10 In what ways can a SolidWorks Flow Simulation project be created?

SolidWorks Flow Simulation project can be created in one of the three ways:

- use the Project Wizard.
- use the template.
- clone an existing project.

11 How do you specify a fluid for a project?

Answer: To specify a fluid for a project, select the fluid from a list of fluids in the SolidWorks Flow Simulation Engineering Database from the **Wizard** (or **General Settings**) dialog box.

12 How does a user define a fluid entering the model with a velocity of 1 m/s?

Answer: Do the following:

- In the Flow Simulation design tree, right-click the `Boundary Conditions` item and select **Insert Boundary Condition**.
- Select the inlet opening's surface.
- Select the **Inlet Velocity** boundary condition type.
- Under **Flow Parameters**, set **Velocity normal to face** to **1 m/s**.
- Click **OK**.

13 The model has a mirror symmetry. Is it OK then to use the Symmetry boundary condition at the model's symmetry plane?

Answer: No. The symmetry condition should be only applied if the flow is symmetric. The geometric symmetry of the model does not always mean that the flow is also symmetric.

14 How do you define a 2D XY plane flow analysis?

Answer: To define a 2D XY plane flow analysis:

- Right-click the `Computational Domain` icon in the SolidWorks Flow Simulation design tree and select **Edit Definition**.
- Click the **Boundary Condition** tab.
- Under **2D plane flow** select **XY - plane flow**.
- Click **OK**.

15 Is it necessary to specify project goals to start the calculation?

Answer: No.

16 How do you start a calculation?

Answer: Click **Flow Simulation**, **Solve**, **Run**, then click **Run**.

17 In the case when you are working with the previously calculated project, what needs to be done first before viewing the result information?

Answer: The first action is to load results.

18 What display features are available in SolidWorks Flow Simulation to view the calculation results?

Answer:

- Cut Plots and Surface Plots (with contours, isolines, vectors)
- 3D-Profile Plots
- XY Plots
- Isosurfaces
- Flow and particles trajectories
- Point, surface and volume parameters
- Goal Plots
- MS Word Reports
- Animation of results

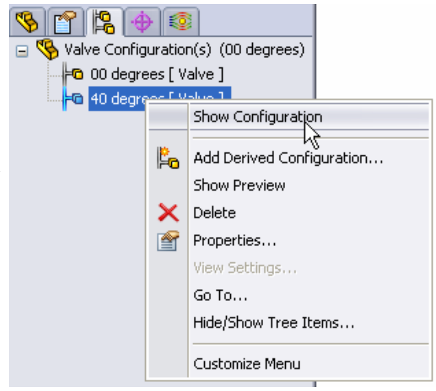
19 How can you calculate the total pressure value for a steady state incompressible fluid?

Answer: For a steady state incompressible fluid the total pressure can be calculated as the sum of static pressure and dynamic head.

20 What is the definition of the total hydraulic resistance (loss) of an obstacle in a pipe?

Answer: It is the difference between the total pressure taken upstream of the obstacle and total pressure taken downstream of the obstacle divided by the incoming dynamic head.

In Class Discussion — Changing the Inlet Boundary Condition



Open the `Valve.SLDPRT` part. Activate the `40 degrees` configuration:

1 Click the ConfigurationManager tab.

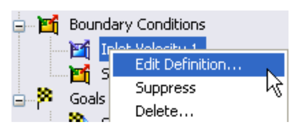
2 In the SolidWorks ConfigurationManager, right-click the `40 degrees` item and select **Show Configuration**.

Ask the students to specify the mass flow rate of 19 kg/s at the inlet opening and calculate the total (including the friction loss) hydraulic loss.

Answer

To specify the inlet mass flow rate of 19 kg/s do the following:

Specify inlet mass flow rate of 19 kg/s



1 In the Flow Simulation Design tree, right-click the `Inlet Velocity 1` icon and select **Edit Definition.**

The **Boundary Condition** dialog box appears.

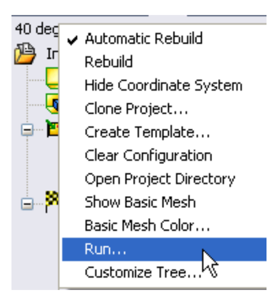
2 In the **Type of Boundary Condition** list, select **Inlet Mass Flow**.

3 Under **Flow Parameters**, set the **Mass Flow Rate Normal to Face** to **19 kg/s**.

4 Click .

With the definition just made, we told SolidWorks Flow Simulation that at this opening 19 kilograms of water per second is flowing into the valve. 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.

Run the analysis and obtain the total hydraulic loss



1 In the Flow Simulation Design tree, right-click the `40 degrees` icon and click **Run**, or click **Flow Simulation, Solve, Run**.

2 Select **New Calculation** and click **Run**. The calculation starts. When calculation is finished, close the **Solver Monitor** dialog box.

Note: To get the new results, you do not need to remesh the model.

3 Right-click the `Goals` icon under the `Results` folder and select **Insert.** The **Goals** dialog box appears.

4 Click **Add All**.

5 Click **OK**. The Excel workbook is created.

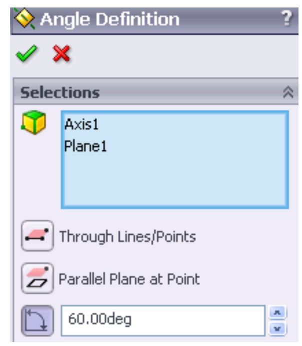
Valve.SLDPRT [40 degrees]

Goal Name	Unit	Value	Averaged Value	Minimum Value	Maximum Value	Progress [%]	Use In Convergence	Delta	Criteria
SG Av Total Pressure Inlet	[Pa]	146224.536	146196.9807	146147.857	146224.536	100	Yes	76.67908563	2261.201662
SG Av Total Pressure Outle	[Pa]	103951.5787	103986.5777	103951.5787	104064.4597	100	Yes	112.8809859	125.5495687
SG Av Dynamic Pressure I	[Pa]	2021.966588	2021.966588	2021.966588	2021.966588	100	Yes	0	2.02197E-05
Hydraulic Loss	[ ]	20.90685254	20.87591517	20.83546984	20.90685254	100	Yes	0.060363803	1.104197034

More to Explore — Modifying the Geometry

Ask the students to change the handle angle to 30 degrees and calculate the total hydraulic loss in this ball valve.

Answer



- Click the FeatureManager tab.
- Right-click the plane item named `Angle Definition` and select **Edit Definition**.
- Set the **At Angle** property to **60**.
- Click.
- Select **Yes** for each message that appears after you click.
- Click the ConfigurationManager tab and rename the `40 degrees` configuration to `30 degrees`.
- Click **Flow Simulation, Solve, Run**. The solver starts.
- After finishing the calculation, click **File, Close** to close the solver monitor dialog box.
- Click the Flow Simulation design tree tab.
- In the SolidWorks Flow Simulation design tree, under `Results`, right-click the `Goals` icon and select **Insert**. The **Goals** dialog box appears.
- Click **Add All**.
- Click **OK**. The Excel workbook is created.

Valve.SLDPRT [30 degrees]

	Unit	Value	Averaged Value	Minimum Value	Maximum Value	Progress [%]	Use In Convergence	Delta	Criteria
SG Average Total Pressure	[Pa]	117098.6995	117114.1858	117091.1756	117131.3438	100	Yes	39.06136648	648.6405339
SG Average Total Pressure	[Pa]	103535.7298	103545.5795	103535.7298	103570.9565	100	Yes	35.22669215	37.99995822
SG Average Dynamic Pres	[Pa]	2009.762517	2009.762517	2009.762517	2009.762517	100	Yes	0	2.00976E-05
Hydraulic loss	[ ]	6.748543456	6.751348097	6.74433195	6.761125604	100	Yes	0.016793655	0.315215609

You can see that the hydraulic loss becomes much lower compared to the `40 degrees` valve.

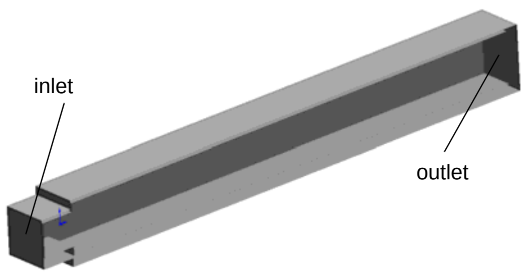
Exercises and Projects — Hydraulic Loss Due to Sudden Expansion

When the fluid passes through ball valve it undergoes two sudden contractions and two sudden expansions. Let us employ SolidWorks Flow Simulation to calculate the hydraulic loss in the simple 2D channel with the sudden expansion.

Tasks

1 Open the `Bilateral expansion channel.sldprt` file in the part located in the corresponding subfolder of the `SolidWorks Curriculum_and_Courseware_2010` folder.

The model is a shell so it is fully closed (the front face on the picture at the right is made transparent to view the results). Therefore, there is no need to create lids.



For easy selection, check that the **Enable selection through transparency** option is enabled under the **Display/Selection** page of the **System Options** dialog box, accessible by clicking **Tools, Options**.

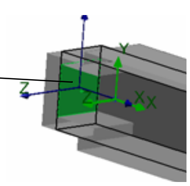
2 Using the **Wizard**, create the SolidWorks Flow Simulation project for internal water analysis with the **Result resolution** level set to **5** (all other settings are default).

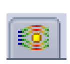
<u>**Answer:**</u> Do the following:

- Click **Flow Simulation, Project, Wizard**.
- Enter a configuration name and click **Next**.
- Click **Next** to accept the default **SI** unit system.
- Click **Next** to accept **Internal** analysis type.
- Select **Water** for **Liquid** and click **Next**.
- Click **Next** to accept the defaults of zero **Roughness** and **Adiabatic wall**.
- Click **Next** to accept the default initial conditions.
- Move the **Result resolution** slider to **5** and click **Finish** to complete the project definition.

3 Specify that water with the velocity of 1 m/s enters the model through the inlet opening. What is the mass flow rate of the incoming water in this case?

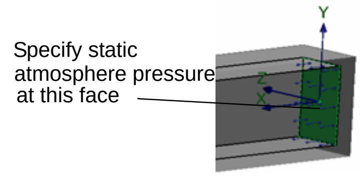
<u>**Answer:**</u> Do the following:



- Click Flow Simulation design tree tab .
- In the Flow Simulation design tree, right-click the `Boundary Conditions` item and select **Insert Boundary Condition**.
- Select the inlet opening face (colored in green).
- Under **Type**, select **Inlet Velocity**.

- Under **Flow Parameters**, enter **1 m/s** for **Velocity Normal to Face** V. Notice that it is equivalent to specifying the mass flow rate of 9.98 kg/s: $\dot{m}$ = ρVA = 998.15*1*0.01.
- Click ✓.

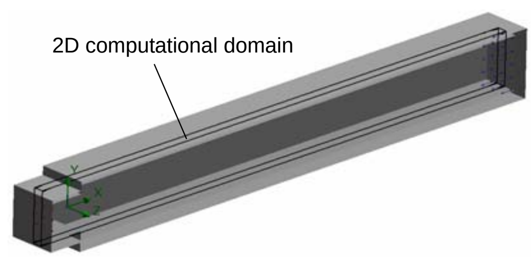
4 Specify that water exits the model through the outlet opening to an area of static atmosphere pressure. What is the value of the ambient static atmosphere pressure in Pa?



Answer: Do the following:

- In the Flow Simulation design tree, right-click the `Boundary Conditions` item and select **Insert Boundary Condition**.
- Select the outlet opening face (colored in green).
- Under **Type**, click **Pressure Openings** and select **Static Pressure**. Notice that the default static pressure value is 101325 Pa - the static atmosphere pressure.
- Click ✓.

5 Specify 2D XY plane flow analysis.



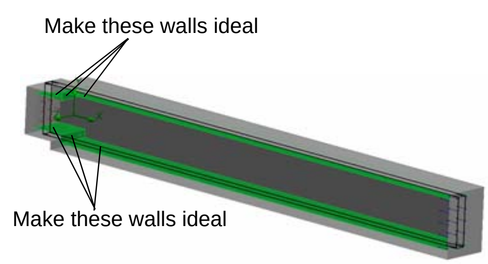
Answer: Do the following:

- In the Flow Simulation design tree, right-click the `Computational Domain` item and select **Edit Definition**.
- Click the **Boundary Condition** tab.
- Under **2D plane flow**, select **XY-Plane flow**.
- Click **OK**.

It is known from hydrodynamics that channels with a sudden expansion generate hydraulic resistance to the flow due to a loss of flow energy caused by vortices in the vortex region downstream of the sudden expansion. Naturally, these regions add to the hydraulic resistance caused by the wall friction as well.

To consider the hydraulic resistance due to the sudden expansion only, let us replace in the calculations the channel's real walls by the "Ideal Walls" boundary condition option in SolidWorks Flow Simulation, which applies adiabatic frictionless walls. As a result, any wall friction will be absent (of course, this can be done in calculations only and it is impossible in physical experiments). The wall friction's influence on the generated vortices, and therefore on the sudden expansion hydraulic resistance, will be neglected for this analysis.

6 Specify the Ideal Wall boundary condition at the channel's walls.



Answer: Do the following:

- In the Flow Simulation design tree, right-click the `Boundary Conditions` item and select **Insert Boundary Condition**.

- Select six channel's walls shown colored in green.
- Under **Type**, click **Wall** and select **Ideal Wall** .
- Click .

7 Specify the Total Pressure and Dynamic Pressure surface goals at inlet.
<u>**Answer:**</u> Do the following:
- In the Flow Simulation Design tree, right-click the `Goals` icon and select **Insert Surface Goal**.
- Click `Inlet Velocity 1` item.
- Under **Parameter**, find **Total Pressure** and **Dynamic Pressure** rows and check the **Av** column for the both.
- Click .

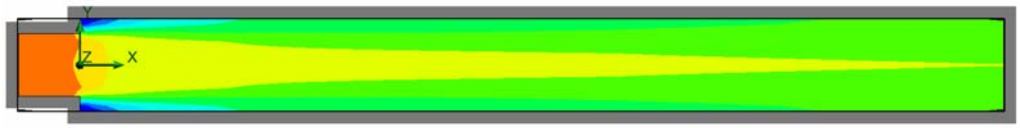
8 Specify the Total Pressure surface goal at outlet.
<u>**Answer:**</u> Do the following:
- In the Flow Simulation Design tree, right-click the `Goals` icon and select **Insert Surface Goal**.
- Click `Static Pressure 1` item.
- Under **Parameter**, find **Total Pressure** and check the **Av** column.
- Click .

9 Specify the Equation goal calculating the total hydraulic loss.
<u>**Answer:**</u> Do the following:
- In the Flow Simulation Design tree, right-click the `Goals` icon and select **Insert Equation Goal**.
- Click the left bracket button .
- In the `Goals` list double-click the `SG Av Total Pressure 1` goal name.
- Click the minus button .
- In the `Goals` list double-click the `SG Av Total Pressure 2` goal name.
- Click the right bracket and the forward slash buttons.
- In the `Goals` list double-click the `SG Av Dynamic Pressure 1` goal name.
- In the **Dimensionality** list select **No units**.
- Click **OK**.

10 Run the calculation.
<u>**Answer:**</u> Do the following:
- Click **Flow Simulation, Solve, Run**.
- Click **Run**. The solver starts.
- When calculation is finished, click **File, Close** in the **Solver Monitor** dialog box.

11 Plot the velocity distribution along the channel.
<u>**Answer:**</u> Do the following:

- Right-click the `Cut Plots` icon and select **Insert**.
- In the **Cut Plot** dialog box, click **View Settings**.
- In the **View Settings** dialog box under **Parameter**, select **Velocity**.
- Click **OK**.
- Click **OK**.



12 Obtain the hydraulic loss caused by the model's sudden expansion by viewing the equation goal average value.

Answer: Do the following:

- Right-click the `Goals` icon and select **Insert**.
- In the **Select Goals** list, select `Equation Goal 1`.
- Click **OK**. The `goals1` Excel workbook is created.
- Switch to `goals1` workbook and see the goal value.

Bilateral expansion channel.SLDPRT [Default (1)]

	Unit	Value	Averaged Value	Minimum Value	Maximum Value	Progress [%]	Use In Convergence	Delta	Criteria
Equation Goal 1	[ ]	0.117284739	0.119232491	0.117284739	0.12329361	100	Yes	0.005307301	0.007415143

Lesson 1 Vocabulary Worksheet – Answer Key

Name________________________________Class: _________ Date:_______________

Fill in the blanks with the proper words.

1 The fluid flow equations solved by SolidWorks Flow Simulation: **time-dependent Reynolds-averaged 3D Navier-Stokes equations employing the k-e turbulence model**

2 The method used for solving these equations with SolidWorks Flow Simulation: **finite volume method**

3 The method used for solving time-independent problems with SolidWorks Flow Simulation: **a steady-state method employing local time steps**

4 The process of subdividing the model into small pieces: **meshing**

5 Splitting mesh cells into smaller ones to better resolve a solid/fluid interface or solution behavior: **mesh refinement**

6 The feature that allows users to track the convergence of a flow parameter(s) in a SolidWorks Flow Simulation project: **goals**

7 The physical feature which must be selected in SolidWorks Flow Simulation to initiate temperature calculation in solids: **heat transfer in solids**

8 The physical feature which must be selected in SolidWorks Flow Simulation to obtain a time-dependent solution: **time dependent**

9 The physical feature which must be selected in SolidWorks Flow Simulation to calculate a flow with significant supersonic regions: **High Mach number flow**

10 The physical feature which must be selected in SolidWorks Flow Simulation to properly calculate a heat convection and/or mixing fluids in low-velocity flows not in weightlessness: **gravitational effects**

11 The physical feature which must be selected in SolidWorks Flow Simulation to fully suppress any flow turbulence in the computational domain: **laminar flow**

12 The SolidWorks Flow Simulation approach of specifying a distributed resistance to a fluid flow: **porous medium**

13 Liquids whose viscosity depends on flow velocity gradients: **non-Newtonian liquids**

Lesson 1 Quiz — Answer Key

Name: ______________________________ Class: _________ Date:_______________

Directions: Answer each question by writing the correct answer or answers in the space provided.

1 What is the specific requirement in SolidWorks Flow Simulation for an internal analyses?

Answer: SolidWorks Flow Simulation internal problem requires that the models be fully closed.

2 What if the fluid used in my design is not defined in the Engineering Database?

Answer: You can specify your own fluid in the Engineering Database.

3 What is the reason for specifying goals in a project?

Answer: Specifying goals as a physical parameter allows you to obtain more reliable results and ensure, that these parameter values convergence within the calculation.

4 Why is it important to specify the proper minimum gap size?

Answer: A properly specified minimum gap size governs the proper cell resolution for small flow passages which will help improve the accuracy of the results.

5 How does a user define a fluid exiting the model at static atmospheric pressure?

Answer: Do the following:

- In the Flow Simulation design tree, right-click the `Boundary Conditions` item and select **Insert Boundary Condition**.
- Select the outlet opening face.
- Under **Basic set of boundary conditions**, select **Pressure openings**.
- Select **Static Pressure**.
- Under the **Settings** tab, specify the **Static pressure** of **101325 Pa** and click **OK**.

6 After obtaining the results you intend to recalculate after changing a boundary condition's value. Do you have to regenerate the computational mesh?

Answer: No. It is not necessary, since the mesh does not change in this case.

7 Can you obtain intermediate results during the calculation?

Answer: Yes, during the calculation you can view section plots of the current results, monitor goal convergence and display minimum and maximum parameter values.

8 How do you load results?

Answer: Right-click the `Results` icon in the Flow Simulation design tree and select **Load Results** or click **Flow Simulation, Results, Load/Unload Results**.

9 You have specified a goal. How can you see the goal value after finishing the calculation?

Answer: To see the goal value after finishing the calculation you need to create a **Goal Plot**:

- Right-click the `Goals` icon in the Flow Simulation design tree and select **Insert**. The **Goals** dialog box appears.

- In the **Goals** dialog box, select the goal name and click **Add**.
- Click **OK**.

10 When can the symmetry condition be applied?

<u>**Answer:**</u> The symmetry condition can be applied if the flow is symmetric.

11 What causes hydraulic losses in a pipeline?

<u>**Answer:**</u> Hydraulic losses are caused by the friction and deformation of the flow stream (due to an obstacle, changing the flow passage, bends, etc.).

Lesson Summary

- SolidWorks Flow Simulation is a fluid flow and heat transfer analysis software fully integrated in SolidWorks.
- Analysis enables you to understand and improve the design by saving time and money by reducing the traditional design cycle.
- SolidWorks Flow Simulation allows you to analyze a wide range of complex problems in fluid mechanics and heat transfer: two- and three-dimensional analyses; external and internal flows; steady-state and transient flows; incompressible and compressible liquid flows; gas flows including subsonic, transonic and supersonic regimes; heat transfer within and between fluids and solids; non-Newtonian liquids (laminar only); laminar, turbulent, and transitional flows; swirling flows and fans; multi-species flows; flows with gravitational effects (also known as buoyancy effects); porous media; fluid flows with solid particles; walls with roughness; surface-to-surface radiation.
- Typical internal analyses, are defined by a fluid which enters a model at the model inlet(s) and exits the model through outlet(s).
- The steps to perform internal fluid flow analysis in SolidWorks Flow Simulation are:
  - Close the model with the lids at inlet and outlet openings.
  - Create a project.
  - Specify boundary conditions.
  - Specify goals.
  - Adjust geometry resolution if necessary.
  - Run the calculation.
  - Obtain the results.
- Goals are physical parameters of interest to the user. Setting goals is a way of conveying to SolidWorks Flow Simulation what you are trying to get out of the analysis, as well as a means of reducing the time SolidWorks Flow Simulation takes to reach a solution. Goals allow you to obtain more reliable results, since you can examine the convergence history of each goal value in the calculation.
- Computational domain is the area where the calculation is performed. Computational domain is divided into small rectangular volumes - cells. This process is called meshing. The mesh cells are subdivided into smaller rectangular cells as necessary to properly resolve the model geometry and the flow features.
- The cell-centered finite volume (FV) method is used to obtain conservative approximations of the governing equations on the locally refined rectangular mesh.