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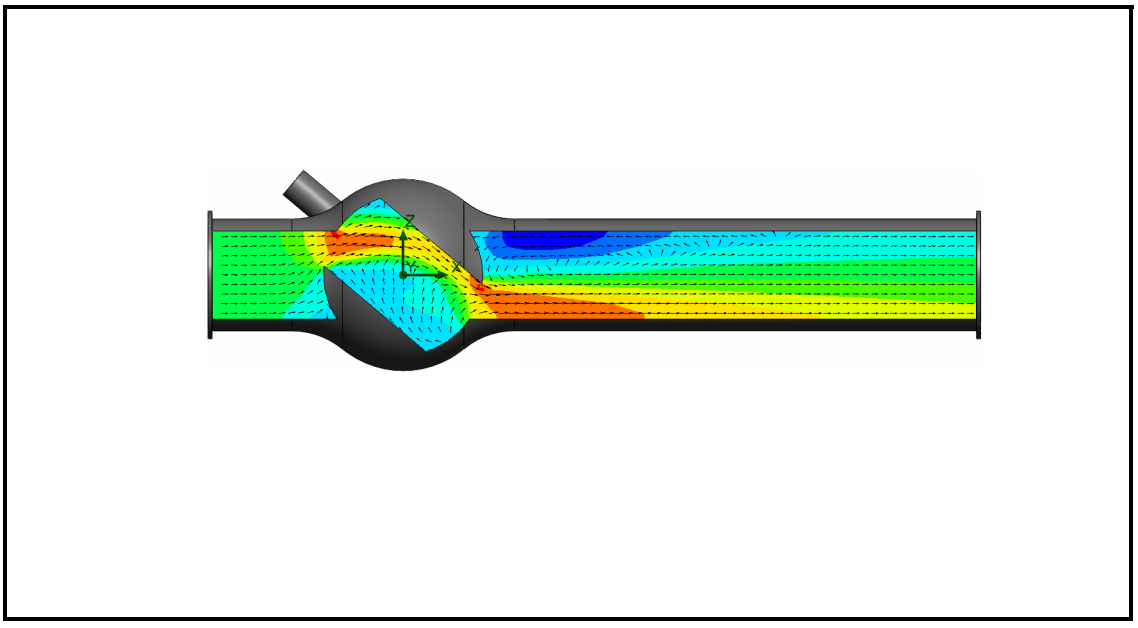
An Introduction to Flow Analysis Applications with SolidWorks Flow Simulation, Student Guide



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Contents

Introduction . . . . . **v**
About This Course v
Online Tutorials v
Course Materials for this Guide v
SolidWorks Simulation Product Line vi

Basic Functionality of SolidWorks Flow Simulation **1**
Goals of This Lesson 1
Active Learning Exercise — Determination of Hydraulic Loss 2
Opening the **`Valve.SLDPRT`** Document 2
Checking the SolidWorks Flow Simulation Menu 2
Model Description 2
Creating Lids Manually 3
Creating Lids Automatically 5
Creating a Project 6
SolidWorks Flow Simulation Design Tree 9
Specifying Boundary Conditions 9
Engineering Goals 10
Specifying Surface Goals 11
Specifying the Equation Goal 12
Running the Calculation 13
Monitoring the Solver 14
Accessing the Results 16
Creating a Cut Plot 17
Displaying Flow Trajectories 19

Creating a Goal Plot 20
Cloning Project 20
Changing the Valve Angle. 21
Changing the Geometry Resolution 22
Changing the Computational Domain. 22
Getting the Valve's Hydraulic Loss. 23
Save Your Work and Exit SolidWorks 23
5 Minute Assessment 24
Project — Hydraulic Loss Due to Sudden Expansion 26
Tasks 26
Lesson 1 Vocabulary Worksheet 29
Lesson Summary 30

i

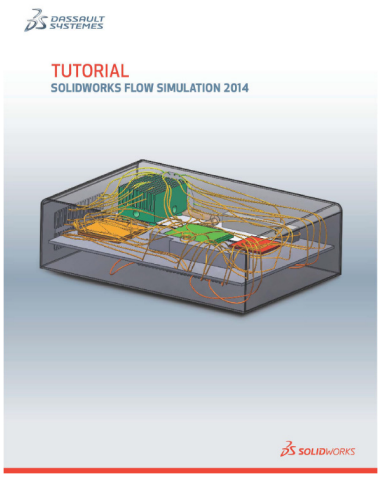
Introduction

About This Course

The *Introduction to Flow Analysis Applications with SolidWorks Flow Simulation* and its supporting materials is designed to assist you in learning SolidWorks Flow Simulation in an academic setting.

Online Tutorials

The *Introduction to Flow Analysis Applications with SolidWorks Flow Simulation* is a companion resource and is supplemented by the SolidWorks Flow Simulation Online Tutorials.

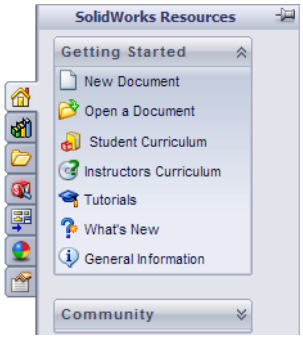


Accessing the Tutorials

To start the Online Tutorials, click **Help, SolidWorks Simulation, Flow Simulation Online Tutorial**. A document will open with the Flow Simulation Tutorials.

Course Materials for this Guide

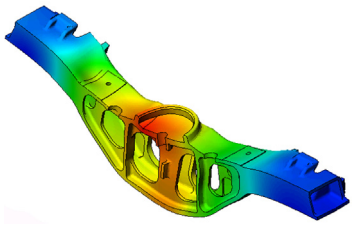
Materials for this guide can be downloaded from within SolidWorks. Click the SolidWorks Resources tab in the Task Pane and then select Student Curriculum. Select the newest Curriculum folder and then click Simulation Student Guides. Control-click the file corresponding to this guide to download the ZIP file. Unzip the file to an accessible location.



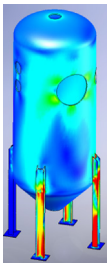
The `EDU_Flow_Simulation_Initial_Model_Files` folder contains the parts needed to complete the lessons. The PDF of the Student Guide can also be found in the ZIP file. Throughout this guide, the `Initial Model Files` folder will be in reference to the above mentioned folder.

SolidWorks Simulation Product Line

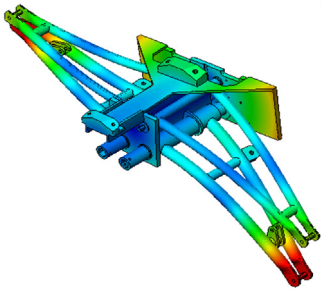
While this course focuses on the introduction to the rigid body dynamics using SolidWorks Motion Simulation, the full product line covers a wide range of analysis areas to consider. The paragraphs below lists the full offering of the SolidWorks Simulation packages and modules.



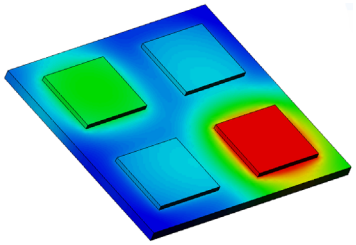
Static studies provide tools for the linear stress analysis of parts and assemblies loaded by static loads. Typical questions that will be answered using this study type are:
Will my part break under normal operating loads?
Is the model over-designed?
Can my design be modified to increase the safety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



Buckling studies analyze performance of the thin parts loaded in compression. Typical questions that will be answered using this study type are:
The 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?

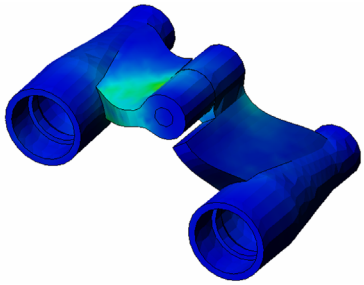


Frequency studies offer tools for the analysis of the natural modes and frequencies. This is essential in the design or many components loaded in both static and dynamic ways. Typical questions that will be answered using this study type are:
Will my part resonate under normal operating loads?
Are the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

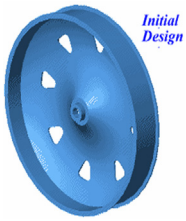


Thermal studies offer tools for the analysis of the heat transfer by means of conduction, convection, and radiation. Typical questions that will be answered using this study type are:
Will the temperature changes effect my model?
How does my model operate in an environment with temperature fluctuation?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

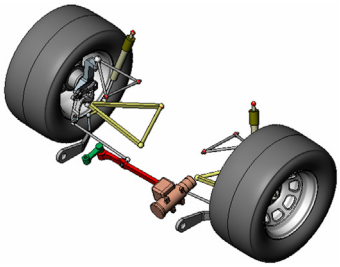
Drop test studies are used to analyze the stress of moving parts or assemblies impacting an obstacle. Typical questions that will be answered using this study type are:
What will happen if my product is mishandled during transportation or dropped?
How does my product behave when dropped on hard wood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

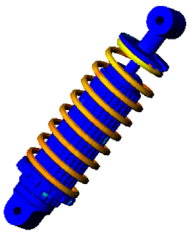


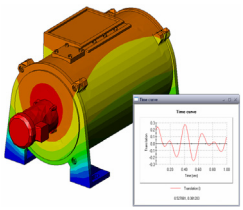
Fatigue studies analyze the resistance of parts and assemblies loaded repetitively over long periods of time. Typical questions that will be answered using this study type are:
Can the life span of my product be estimated accurately?
Will modifying my current design help extend the product life?
Is my 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?



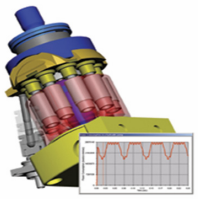
Pressure Vessel Design studies combine the results of static studies with the desired factors. Typical questions that will be answered using this study type are:
Is my pressure vessel design safe?
How do you optimize the thickness of your pressure vessel design?

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 a given load?
Does my model experience excessive bending during normal operating conditions?

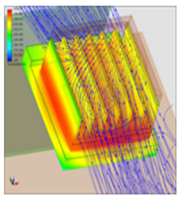




Dynamic studies analyze objects forced by loads that vary in time. Typical examples could be shock loads of components mounted in vehicles, turbines loaded by oscillatory forces, aircraft components loaded in random fashion, etc. Both linear (small structural deformations, basic material models) and 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 the vehicle hits a large pothole on the road, designed safely? How much does it deform under such circumstances?



Motion Simulation enables users 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 displacements, velocities, and accelerations of the mechanism components?
Is the mechanism efficient? Can it be improved?



Flow Simulation enables users to analyze the behavior and effect of moving fluids around or within parts and assemblies. Heat transfer in both fluids and solids is considered as well. Pressure and temperature effect can subsequently be transferred into SolidWorks Simulation studies to continue with the stress analysis. Typical questions that will be answered using this modulus are:
Is the fluid moving too fast and will it cause problems in my design?
Is the moving fluid too hot or too cold?
Is the heat transfer in my product efficient? Can it be improved?
How effective is my design at moving fluid through the system?

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?

Along with the packages stated above there are separate packages that perform other types of simulations. These include:

- SolidWorks Flow Simulation
- SolidWorks Plastics
- SolidWorks Sustainability

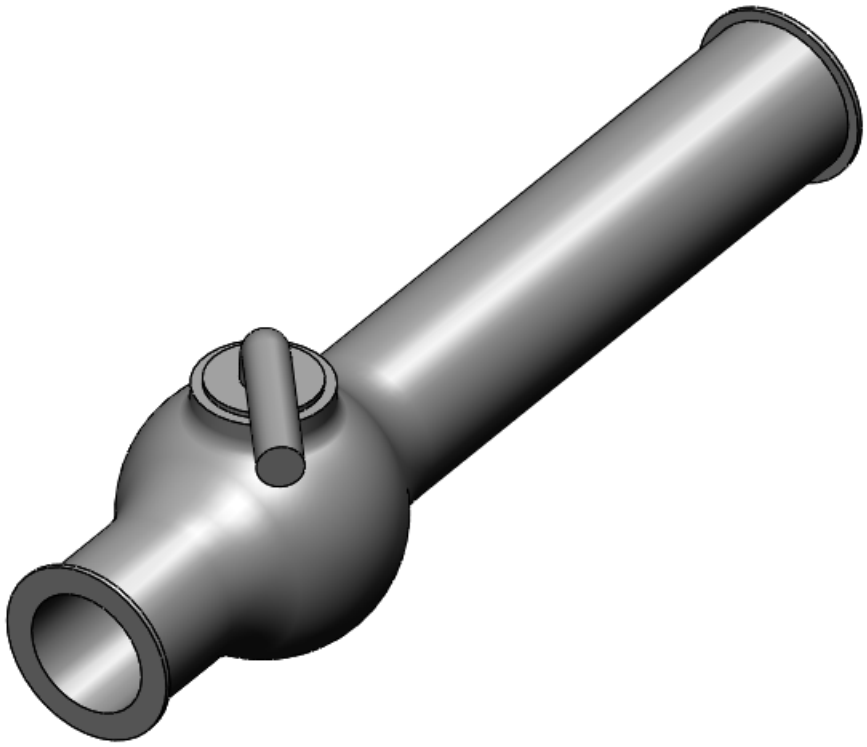
There are guides for instructors and students covering SolidWorks Simulation as well as SolidWorks Motion. Projects for various topics such as those mentioned above can be found in the student or instructor curriculum.

1

Basic Functionality of SolidWorks Flow Simulation

Goals of This Lesson

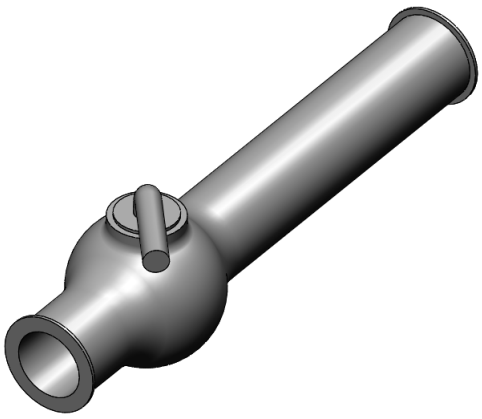
Upon successful completion of this lesson, you will be able to understand the basic functionality of SolidWorks Flow Simulation and perform hydraulic analysis on the following part.



Active Learning Exercise — Determination of Hydraulic Loss

Use SolidWorks Flow Simulation to perform internal fluid analysis on the `Valve.SLDPRT` part shown to the right.

The step-by-step instructions are given below.

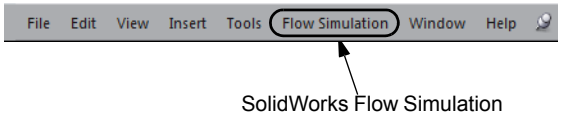


Opening the `Valve.SLDPRT` Document

1 Click **File**, **Open**. In the **Open** dialog box, browse to the `Valve.SLDPRT` part located in the `Initial Model Files` folder and click **Open** (or double-click the part).

Checking the SolidWorks Flow Simulation Menu

If SolidWorks Flow Simulation is added-in, 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**.

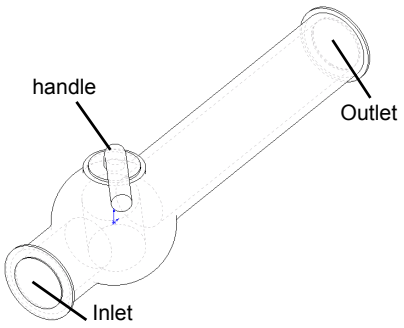
If SolidWorks Flow Simulation is not in the list, you need to install SolidWorks Flow Simulation first.

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 the 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.

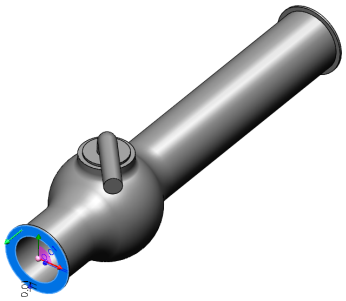
Note: Internal flow analyses are analyses where fluid enters a model at the inlets and exits the model through the outlets. The exception are some natural convection problems that may not have openings.

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 automatically; both of the procedures are shown below.

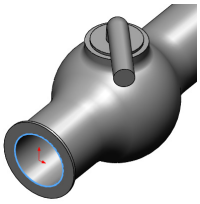
Creating Lids Manually

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. The entity will automatically be converted. If not, click in the **Convert Entities** PropertyManager.
5. Complete the sketch by clicking the **OK** button in the confirmation corner of the graphics area.

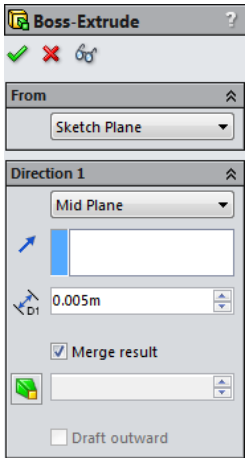


6 Click **Extruded Boss/Base** on the `Features` toolbar.

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.

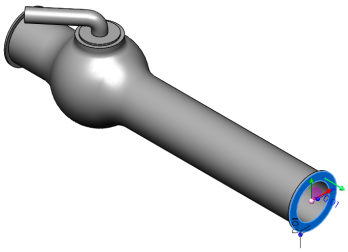


Next, in the same manner we will create the outlet lid.

Creating Outlet Lid

1 Select the face shown in the picture.

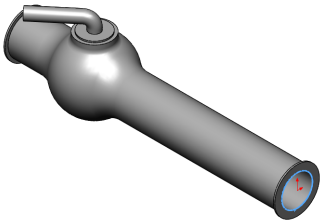
2 Click **Sketch** on the `Sketch` toolbar.

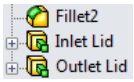


3 Select the tube's inner edge.

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

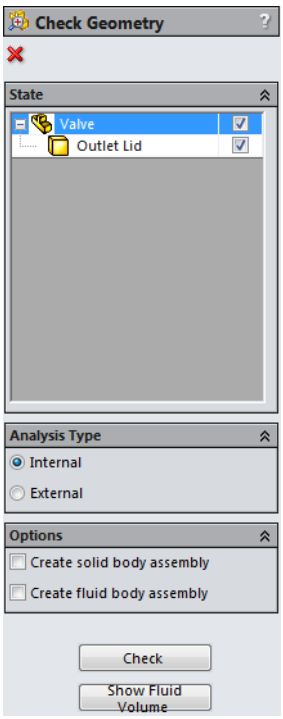
5 Rename the new extrusions `Extrude1` and `Extrude2` to `Inlet Lid` and `Outlet Lid`, correspondingly.





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. Select **Internal** under **Analysis Type.**
3. 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

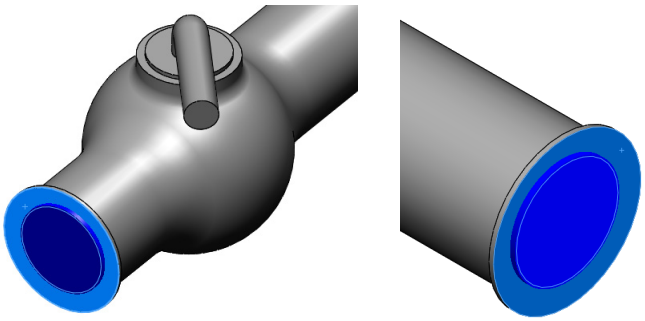
The previous step showed the manual lid creation. In the next step you will practice the SolidWorks Flow Simulation automatic lid creation tool. This tool can save considerable amount of time if multiple lids are needed to close the internal volume.

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.



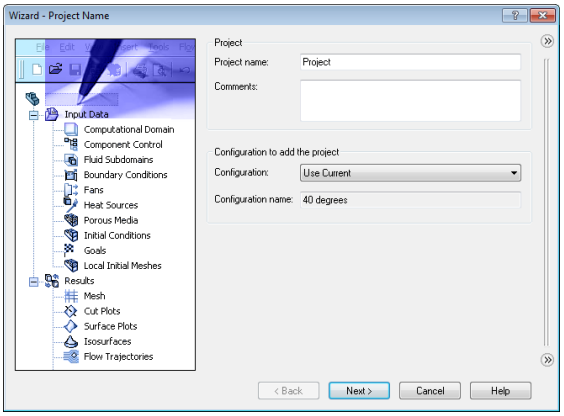
Note: In the assembly mode, each newly created lid forms a new part saved in the assembly folder.

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.

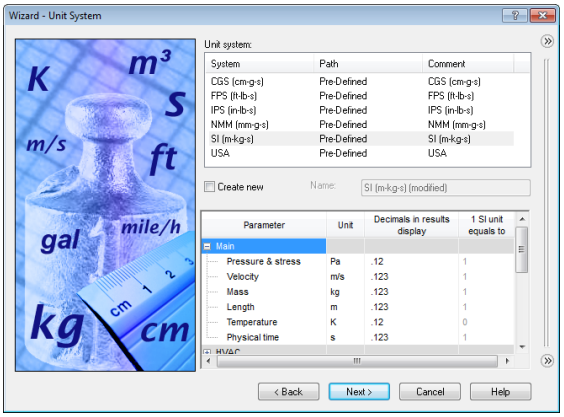
2 In the **Project Configuration** dialog box, click **Use current** (`40 degrees`).



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**.

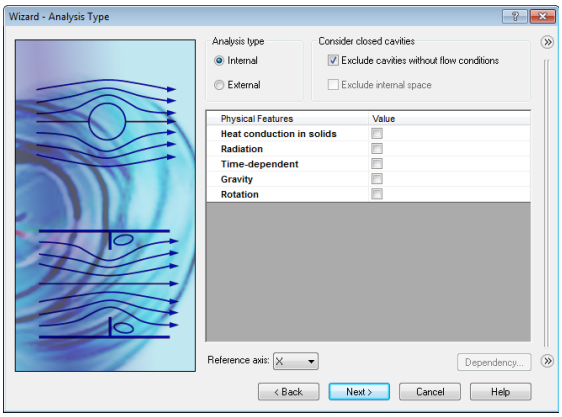
3 In the **Unit System** dialog box, you can select the desired system of units for both input and output (results).



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

Click **Next**.

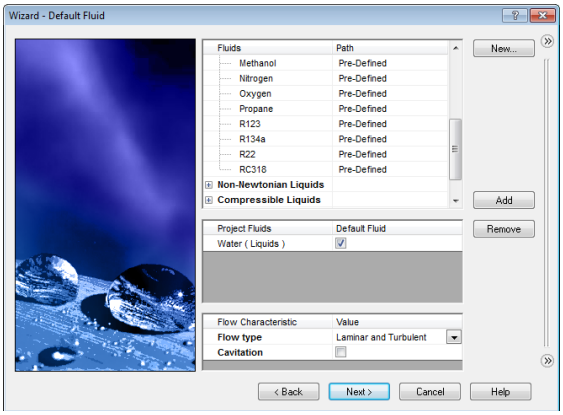
4 In the **Analysis Type** dialog box you can select either **Internal** or **External** type of the flow analysis. This dialog also allows you to specify advanced physical features you want to take into account: heat transfer in solids, surface-to-surface radiation, time-dependent effects, gravity and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

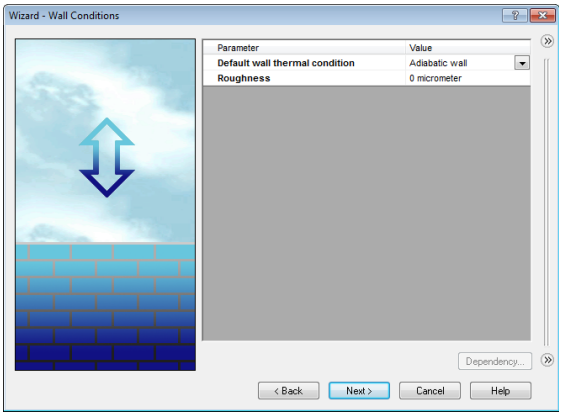


Note: The SolidWorks Flow Simulation **Engineering Database** contains physical properties of predefined and user-defined gases, real gases, incompressible liquids, non-Newtonian liquids, compressible liquids, solid substances and porous materials. It includes both constant values and tabular dependencies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

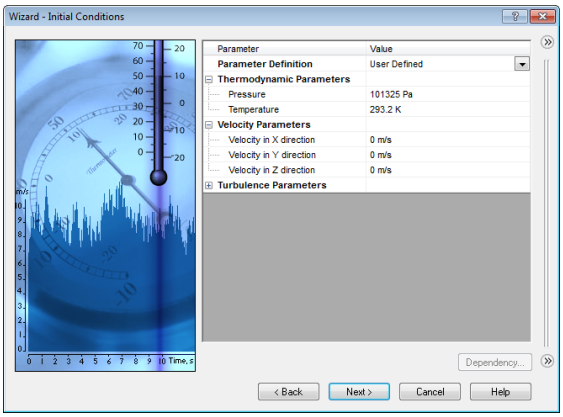
In this project we will not deal with the rough walls and heat conduction through the walls, so leave the default settings and click **Next**.



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.

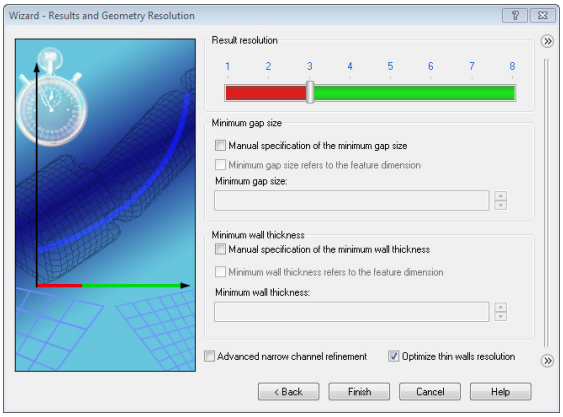
For this project use the default values.

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).

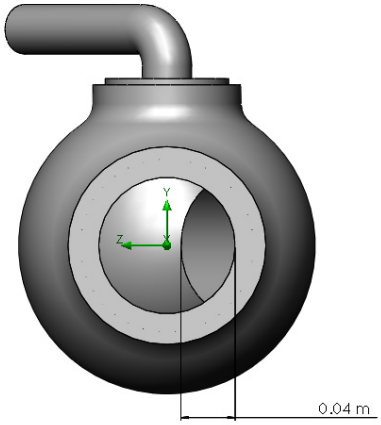


For this project accept the default **Result resolution** level **3**.

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.

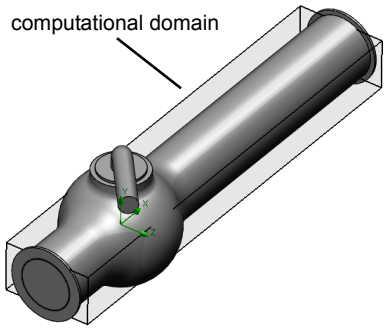
Click **Finish**.

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 DisplayManager 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.



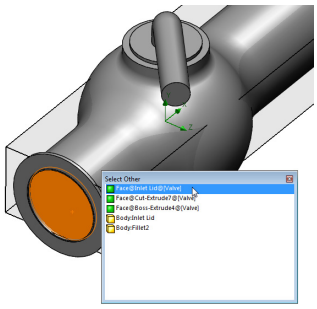
Now let us specify the other parts of the project.

The next step is the specification 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 or move the mouse over the faces in the list until the inner face is highlighted, then click the left mouse button.

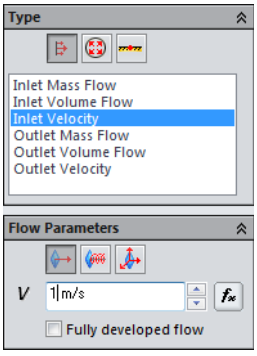
   The selected face appears in the **Faces to Apply the Boundary Condition** list.



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** to **1 m/s** (just type the value, the units will appear automatically).

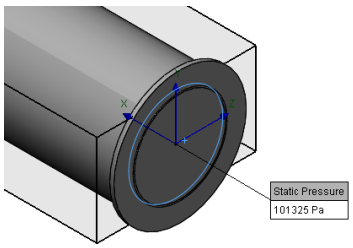
   Accept all other parameters and click .

   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 In the SolidWorks Flow Simulation design tree, right-click the `Boundary Conditions` icon select **Insert Boundary Condition**. The **Boundary Condition** PropertyManager appears with the selected face in the **Faces to Apply the Boundary Condition** list.

Select the `Outlet Lid` inner face.

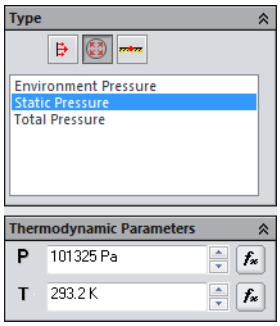


Let us specify pressure on this boundary, otherwise the problem specification is deficient. Before the calculation starts, SolidWorks Flow Simulation checks the specified boundary conditions for mass flow rate balance. The specification of boundary conditions is incorrect if the total mass flow rate 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 .



Engineering Goals

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 m/s) and the water density (998.1934 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.

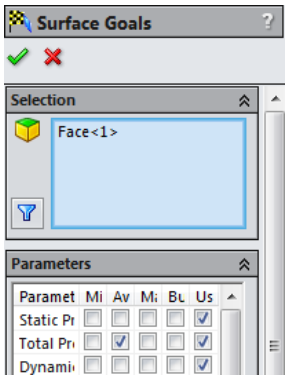
Engineering goals are the parameters which the user is interested in. Setting goals is essentially a way of conveying to SolidWorks Flow Simulation what you are trying to get out of the analysis, as well as means of reducing the time SolidWorks Flow Simulation takes to reach a solution. By only selecting 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**.