COSMOS Student Workbook

**Contents**

Lesson 1: Basic Functionality of COSMOSWorks

Upon successful completion of this lesson, you will be able to understand the basic functionality of COSMOSWorks and perform static analysis to the following assembly.
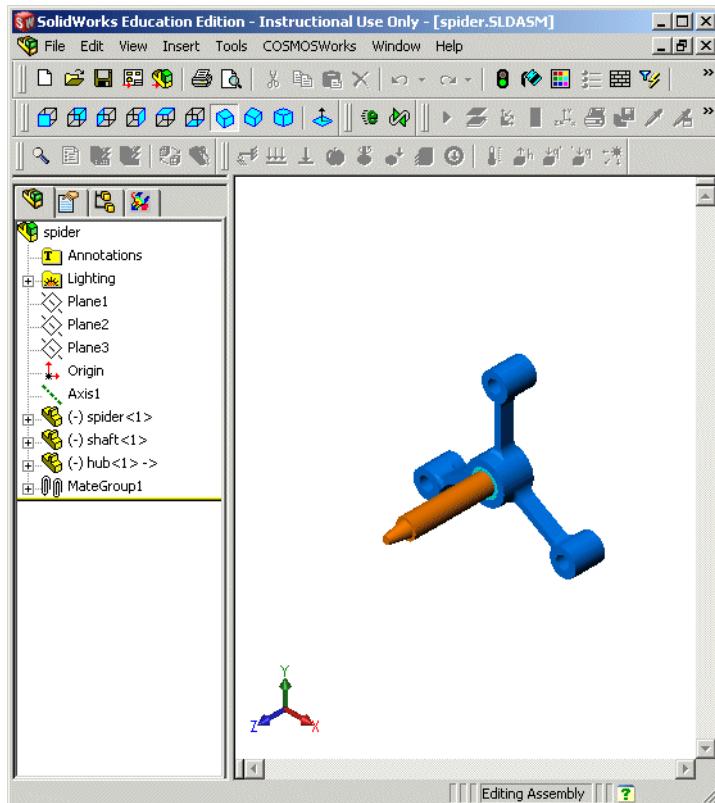
Active Learning Exercise — Performing Static Analysis

Use COSMOSWorks to perform static analysis on the `Spider.SLDASM` assembly shown to the right.

The step-by-step instructions are given below.
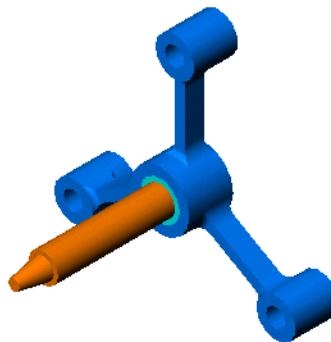
Opening the Spider.SLDASM Document

1 Click **Open** 📂 on the Standard toolbar. The **Open** dialog box appears.

2 Navigate to the **Examples** folder in the COSMOSWorks' installation directory.

3 Select `Spider.SLDASM`.

4 Click **Open**.
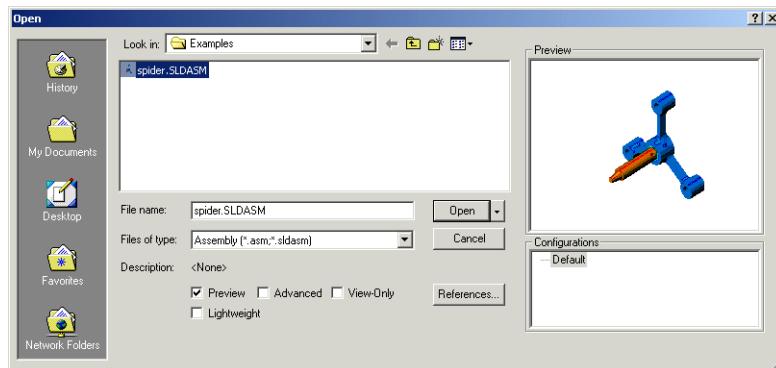
The `spider.SLDASM` assembly opens.

The `spider` assembly has three components: the shaft, hub, and spider leg. The following exploded view of the assembly shows these components.
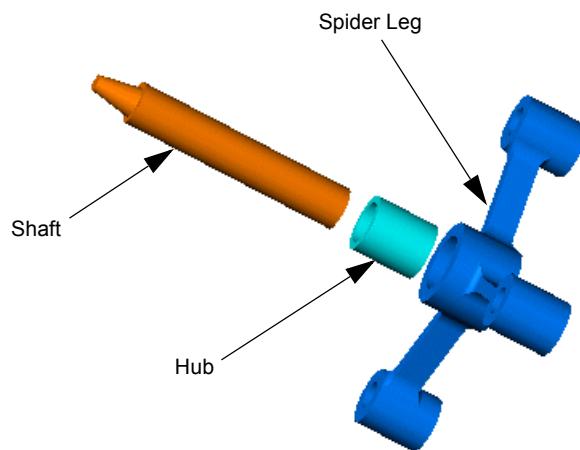
Checking the COSMOSWorks Menu

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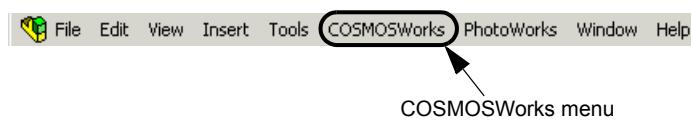
COSMOSWorks menu

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

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

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

3 Click **OK**.

The `COSMOSWorks` menu appears on the SolidWorks' menu bar. Also, the COSMOSWorks Manager tab 📊 appears at the top of the left pane.

Saving the Assembly to a Temporary Directory

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

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

2 Click **File**, **Save As**.

The **Save As** dialog box appears.

3 Click **References**.

The **Edit Referenced File Locations** dialog box appears.
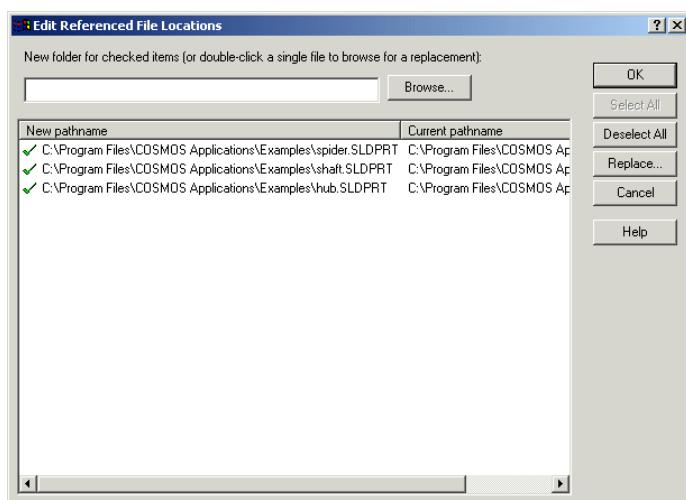
4 Check the three part files named `spider.SLDPRT`, `shaft.SLDPRT`, and `hub.SLDPRT`.

5 Click **Browse**. The **Browse for Folder** dialog box appears.
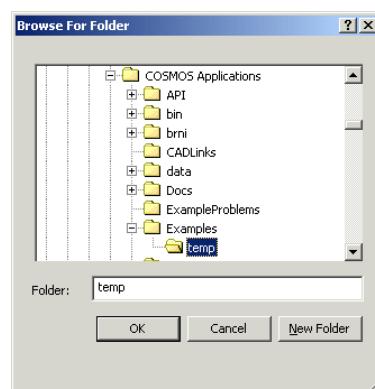
6 Navigate to the **temp** directory created in step 1 and click **OK**. The full path to the new location of the assembly and the associated part files appears at the top of the `Edit Referenced File Locations` dialog box as shown in the next figure.
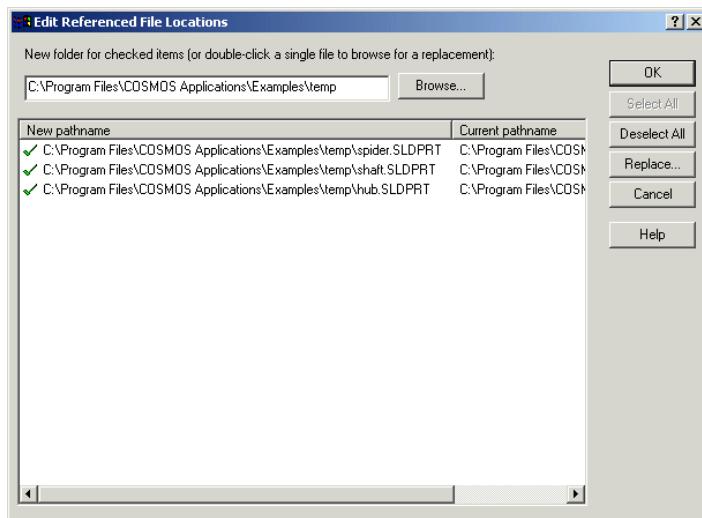
7 Click **OK**.

8 In the **Save As** dialog box, navigate to the `temp` directory and click **Save** to save the assembly file `spider.SLDASM`. SolidWorks creates another copy of the assembly file along with associated part files in the `temp` directory.

Switching to COSMOSWorks Manager

Switch to COSMOSWorks by clicking the COSMOSWorks Manager tab 🗷 at the top of the left pane.

Setting the Analysis Units

Before we start this lesson, we will set the analysis units.

1 Click **COSMOSWorks**, **Preferences**.

The **Preferences** dialog box appears.

2 Click the **Units** tab.

3 Click **English (IPS)** in the **System of units** list.

4 Click **in** in the **Length units** list.

5 Click **OK**.
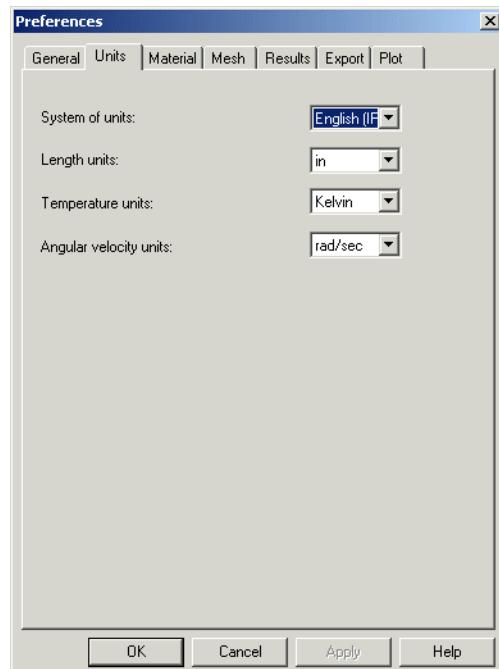
Step 1: Creating a Study

The first step in performing analysis is to create a study.

1   Right-click the `spider` icon and click **Study**, or click **COSMOSWorks**, **Study**.

The **Study** dialog box appears.

2   Under **Study name**, type `My First Study`.

3   Under **Analysis type**, select **Static**.

4   Under **Mesh type**, select **Solid mesh**.

5   Click **OK**.

COSMOSWorks creates a representative tree for the study in the COSMOSWorks Manager.
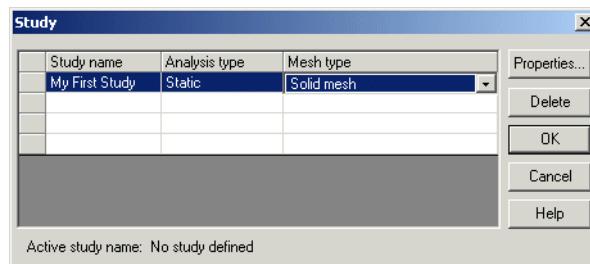
Step 2: Assigning Material

All assembly components are made of Alloy Steel.
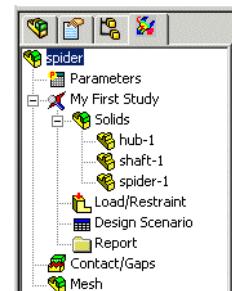
Assign Alloy Steel to All Components

1   In the COSMOSWorks Manager tree, right-click the **Solids** folder and click **Apply Material to All**.

The **Material** dialog box appears.

2   Under **Select material source**, do the following:

a)   Click **From library files**.

b)   Select **coswkmat** from the menu.

c)   Click the plus sign ⊞ next to the **Steel** material category and select **Alloy Steel**.
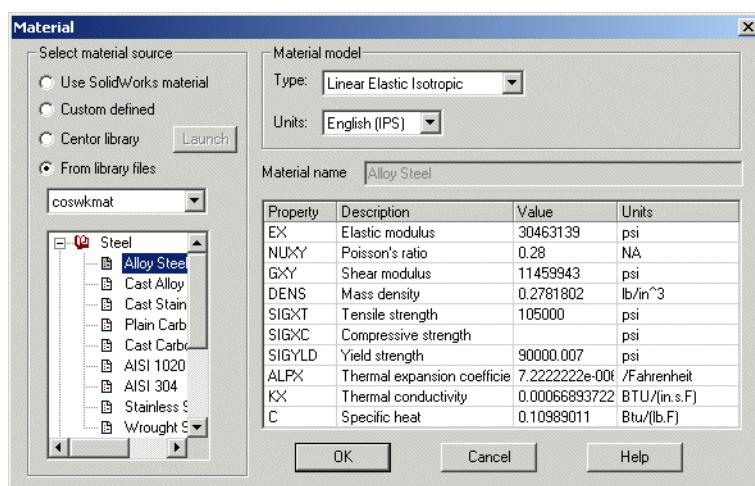
Note: The mechanical and physical properties of Alloy Steel appear in the table to the right.

---

3   Click **OK**.

Alloy steel is assigned to all components of the assembly and a check mark appears on each component's icon. Note that the name of the assigned material appears next to the component's name.
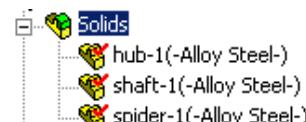
Step 3: Applying Restraints

We will fix the three holes.

1 Use the **Arrow** keys to rotate the assembly as shown in the figure.
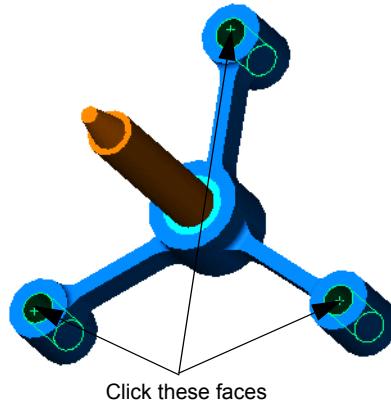
Click these faces

2 In the COSMOSWorks Manager tree, right-click the **Load/Restraint** folder and click **Restraints**.

The **Restraint** PropertyManager appears.

3 Make sure that **Type** is set to **Fixed**.

4 In the graphics area, click the faces of the three holes.

Face<1>, Face<2>, and Face<3> appear in the **Selected entities** list box.

5 Click **OK** ✔.

The Fixed restraint is applied and its symbols appear on the selected faces.

Fixed Restraint symbols
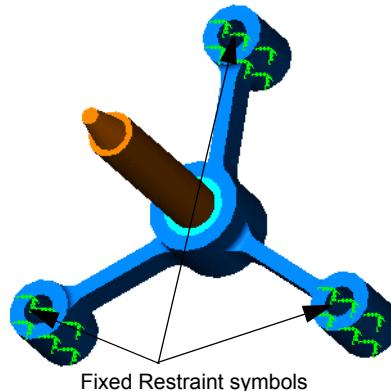
Also, a restraint icon (Restraint-1) appears in the Load/Restraint folder in the COSMOSWorks Manager tree.

Step 4: Applying Loads

We will apply a 500-lb. force normal to the face shown in the figure.

1 Click **Zoom to Area** ⊕ on the View toolbar and zoom into the tapered part of the shaft.

2 In the COSMOSWorks Manager tree, right-click the **Load/Restraint** folder and click **Force**.

The **Force** PropertyManager appears.

3 Under **Type**, click **Apply normal force**.

4 In the graphics area, click the face shown in the figure.

Face<1> appears in the **Selected entities** list box.

5 Make sure that **Units** is set to **English (IPS)**.

6 In the **Normal Force/Torque (Per entity)** ↓ box, type **500**.

7 To view the force before it is applied, click **Preview** 👓 .

8 Click **OK** ✔ .

COSMOSWorks applies the force to the selected face and a Force-1 icon appears in the Load/Restraint folder.

Click this face
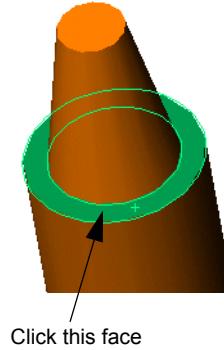
To Hide Restraints and Loads Symbols

In the COSMOSWorks Manager tree, right-click the **Load/Restraint** folder and click **Hide All**.

Step 5: Meshing the Assembly

Meshing divides your model into smaller pieces called elements. COSMOSWorks suggests an element size for convenience.

1 In the COSMOSWorks Manager tree, right-click the **Mesh** icon and click **Create**.

The **Mesh** PropertyManager appears.

2 Click **Preferences**.
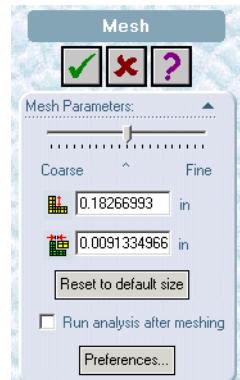
The **Preferences** dialog box appears with the **Mesh** tab selected.

3 Under **Mesh quality**, click **High**.

4 Under **Mesher type**, click **Standard**.

5 Select **4 Points** from the **Jacobian check** menu.

6 Under **Mesh Control**, click **Smooth Surface**. Make sure that **Automatic transition** is not checked.

7 Click **OK**.

You will return to the **Mesh** PropertyManager.

8 Accept the default **Global Size** and **Tolerance** suggested by the program and click **OK** .

Meshing starts. When meshing is completed, a check mark appears on the Mesh icon and the mesh is displayed in the graphics area.

Global size

Global element size is a measure of the average edge length of elements in a mesh
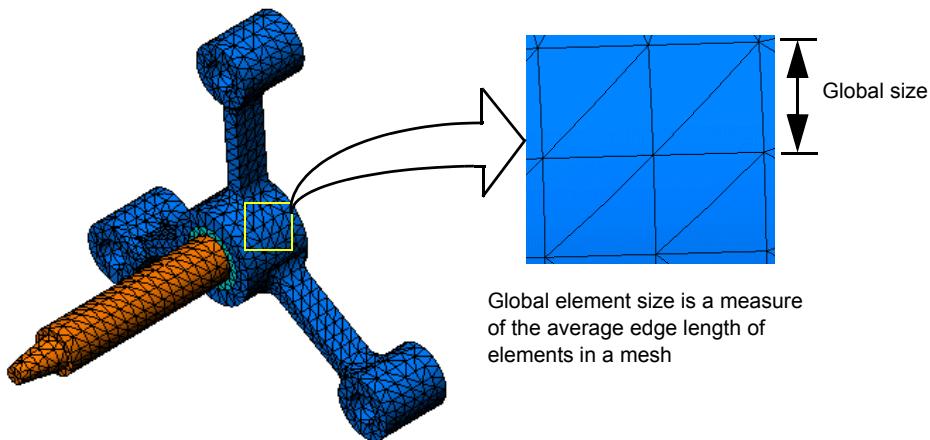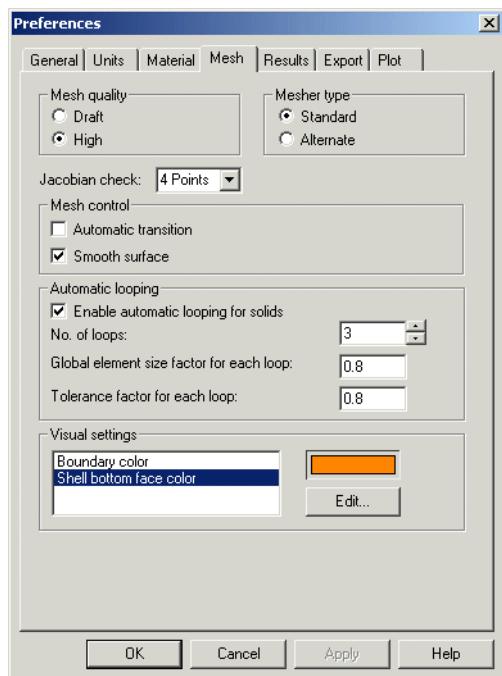
Step 6: Running the Analysis

In the COSMOSWorks Manager tree, right-click the **My First Study** icon and click **Run**.

Analysis starts. When analysis is completed, COSMOSWorks creates result folders automatically in the COSMOSWorks Manager tree.

Step 7: Visualizing the Results

von Mises stress

1 Click the plus sign ⊞ beside the **Stress** folder.

Plot1 icon appears.

2 Double-click **Plot**.

Von Mises stress plot is displayed.
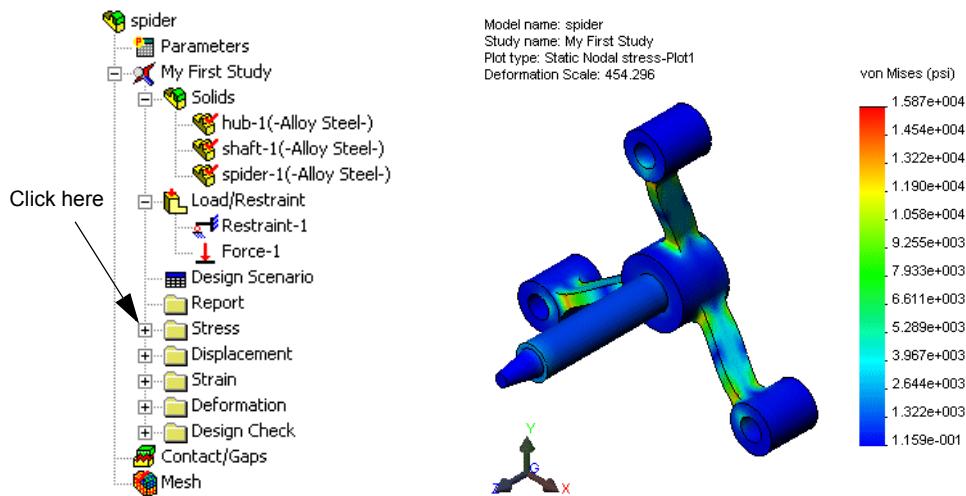
Animating the Plot

1 Right-click **Plot1** and click **Animate**.

The **Animation** dialog box appears.

2 Check **Save as AVI File**, then click **Browse** and select a destination folder to save the AVI file.

3 Click ▶ to play the animation.

4 Click ■ to stop the animation.

5 Click ⊠ to close the **Animation** dialog box.
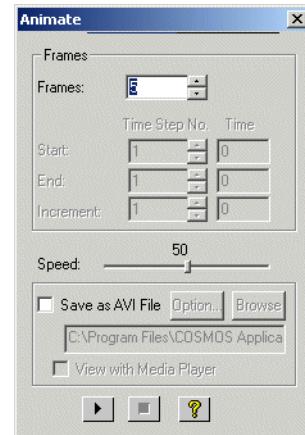
Visualizing Resultant Displacements

1 Click the plus sign ⊞ beside the **Displacement** folder.

Plot1 icon appears.

2 Double-click **Plot1**.

Resultant displacement plot is displayed.
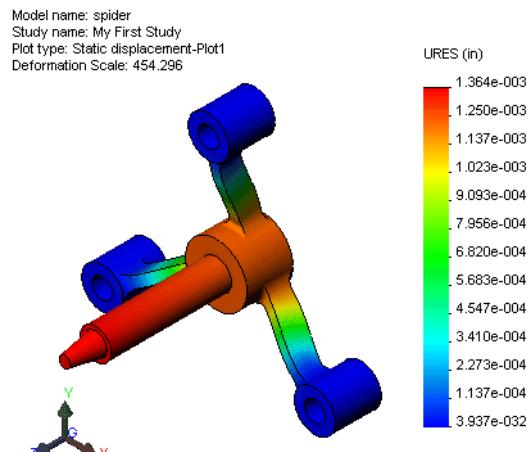
Is the Design Safe?

The **Design Check Wizard** can answer this question for you. We will use the wizard to estimate the factor of safety at every point in the model. In the process, you need to select a failure criterion.

1   Click the plus sign ⊞ beside the **Design Check** folder.

Plot1 appears.

2   Right-click the **Plot1** icon and select **Edit Definition**.

Design Check Wizard Step 1 of 3 dialog box appears.

3   Under **Criterion**, click **Maximum von Mises stress**.

> **Note:** Several criteria are available. The von Mises criterion is usually used to check the failure of ductile materials.

4   Click **Next**.

Design Check Wizard Step 2 of 3 dialog box appears.

5   Under **Set stress limit**, click **to Yield strength**.

> **Note:** When a material yields, it continues to deform even if the load is not increased.

6   Click **Next**.

Design Check Wizard Step 3 of 3 dialog box appears.

7   Under **Plot results**, click **Areas below factor of safety** and enter **1**.

8   Click **Finish**. The plot is generated.
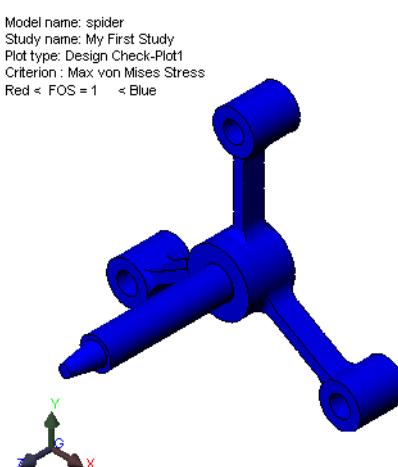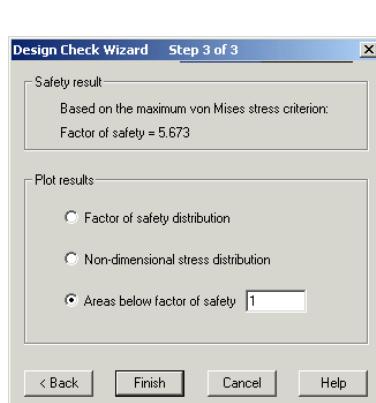
Rotate the model and look for unsafe areas shown in red. The plot is free from the red color indicating that all locations are safe.

How Safe is the Design?

1 Right-click the **Design Check** folder in the analysis tree and click **Define**.

Design Check Wizard Step 1 of 3 dialog box appears.

2 Click **Next**.

Design Check Wizard Step 2 of 3 dialog box appears.

3 Click **Next**.

Design Check Wizard Step 3 of 3 dialog box appears.

4 Under **Plot results**, click **Factor of safety distribution**.

5 Click **Finish**.

Model name: spider
Study name: My First Study
Plot type: Design Check-Plot2
Criterion : Max von Mises Stress
Factor of safety distribution: Min FOS = 5.7

FOS

1.000e+002
9.214e+001
8.428e+001
7.642e+001
6.856e+001
6.070e+001
5.284e+001
4.498e+001
3.712e+001
2.925e+001
2.139e+001
1.353e+001
5.673e+000

The generated plot shows the distribution of the factor of safety. The smallest factor of safety is approximately 5.7.
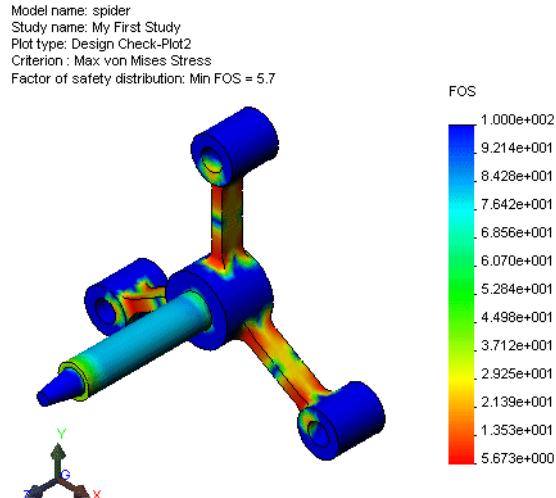
Note: A factor of safety of 1.0 at a location means that the material is just starting to yield. A factor of safety of 2.0, for example, means that the design is safe at that location and that the material will start yielding if you double the loads on the model

Saving All Generated Plots

Right-click **My First Study** icon and click **Save all plots as JPEG files**.

The program saves all result plots in the following directory: "`\work\reports\spider-My First Study`" inside the COSMOSWorks' installation directory.

Generating a Study Report

The **Report** utility helps you document your work quickly and systematically for each study. The program generates structured, Internet-ready reports (HTML files) that describe all aspects related to the study.

1 Right-click the **Report** icon and click **Define**.

The **Report** dialog box appears.

The **Settings for** list box lists the sections to be included in the report. To include a section, make sure that it is checked. To exclude a section, clear it from the list.

2 In the **Logo File** box, browse to the location of your company's logo file. Admissible formats are: **JPEG Files (*.jpg)**, **GIF Files (*.gif)**, or **Bitmap Files (*.bmp)**.

3 In the **Title** box, type a title for your report.

4   Supply the rest of information (**Author**, **Company**, and **Date**) in their respective fields.

5   Select the **Show report on OK** check box.

6   To edit the contents of a section, highlight it in the **Settings for** box.

The **Preview** box lets you enter the desired information in the report.

7   Highlight **Conclusion** in the **Settings for** list and enter a conclusion of your study in the box to the right.

8   Click **OK**.

The report opens in your default internet browser.

Also, the program creates an icon in the Report folder in the COSMOSWorks Manager tree.

To edit any section of the report, right-click the report icon and click **Edit Definition**. Modify the section and click **OK** to replace the existing report.
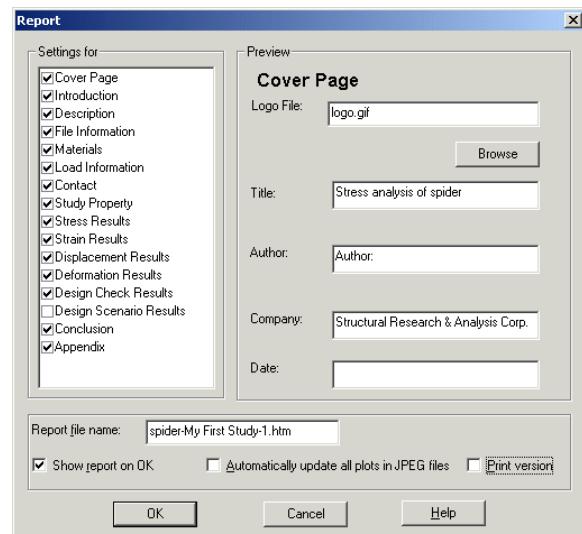
Save Your Work and Exit SolidWorks

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

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

5 Minute Assessment

1 How do you start a COSMOSWorks session?

_____

_____

2 What do you do if COSMOSWorks menu is not on the SolidWorks' menu bar?

_____

_____

3 What types of documents can COSMOSWorks analyze? _____

_____

4 What is analysis? _____

_____

5 Why analysis is important?_____

_____

6 What is an analysis study? _____

_____

7 What types of analysis COSMOSWorks can perform? _____

_____

8 What does static analysis calculate?_____

_____

9 Why static analysis is important?_____

_____

10 What is stress? _____

_____

11 What are the main steps in performing analysis?_____

_____

12 How can you change the material of a part? _____

_____

13 What tool do you use to check the safety of your design? _____

_____

14 The Design Check Wizard shows a factor of safety of 0.8 at some locations. Is your design safe? _____

_____

Projects — Deflection of a Beam Due to an End Force

Some simple problems have exact answers. One of these problems is a beam loaded by a force at its tip as shown in the figure. We will use COSMOSWorks to solve this problem and compare its results with the exact solution.

Tasks

1 Open the `cantilever.sldprt` file located in the `Examples` folder of the COSMOSWorks installation directory.

2 Measure the width, height, and length of the cantilever.

3 Save the part to another name.

4 Switch to the COSMOSWorks Manager tree.

5 Create a static study.

6 Assign `Alloy Steel` to the part. What is the value of the elastic modulus in psi?

Answer: _____

7 Fix one of the end faces of the cantilever.

8 Apply a downward force to the upper edge of the other end face with magnitude of 100 lb.

9 Mesh the part and run the analysis.

10 Plot the displacement in the Y-direction. The Y-direction is the same as dir 2 of Plane1. What is the maximum Y-displacement at the free end of the cantilever?

Answer: _____

11 Calculate the theoretical vertical displacement at the free end using the following relation:

$$UY_{Theory} = \frac{4FL^3}{Ewh^3}$$

Answer: _____

_____

12 Calculate the error in the vertical displacement using the following relation:

$$ErrorPercentage = \left(\frac{UY_{Theory} - UY_{COSMOS}}{UY_{Theory}}\right)100$$

Answer: _____
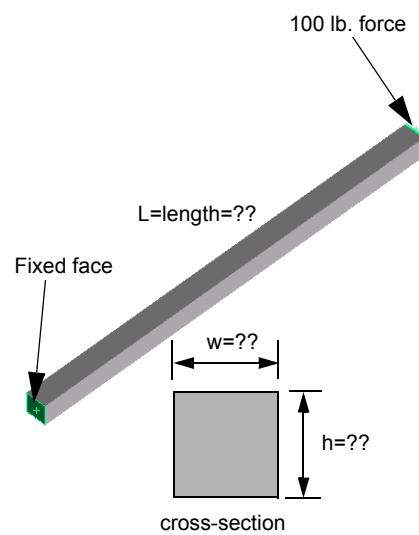
_____

_____

Lesson 1 Vocabulary Worksheet

Name_____Class: _____ Date:_____

Fill in the blanks with the proper words.

1 The sequence of creating a model in SolidWorks, manufacturing a prototype, and testing it: _____

2 A *what-if* scenario of analysis type, materials, restraints, and loads: _____

3 The method that COSMOSWorks uses to perform analysis: _____

4 The type of study that calculates displacements, strains, and stresses: _____

5 The process of subdividing the model into small pieces: _____

6 Small pieces of simple shapes created during meshing: _____

7 Elements share common points called: _____

8 The force acting on an area divided by that area: _____

9 The sudden collapse of slender designs due to axial compressive loads: _____

10 A study that calculates how hot a design gets: _____

11 A number that provides a general description of the state of stress: _____

12 Normal stresses on planes where shear stresses vanish: _____

13 The frequencies that a body tends to vibrate in: _____

14 The type of analysis that can help you avoid resonance: _____

Lesson 1 Quiz

Name: _____Class: _____ Date:_____

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

1 How do you switch between the COSMOSWorks Manager and the FeatureManager? _

_____

2 You test your design by creating a study. What is a study? _____

_____

3 What types of analyses can you perform in COSMOSWorks? _____

_____

4 After obtaining the results of a study, you changed the material, loads, and/or restraints. Do you have to mesh again?_____

_____

5 After meshing a study, you changed the geometry. Do you need to mesh the model again? _____

_____

6 How do you create a new static study? _____

_____

7 What is a mesh? _____

_____

8 In an assembly, how many icons you expect to see in the Solids folder? _____

Lesson 2: Basic Functionality of COSMOSFloWorks

Goals of This Lesson

Upon successful completion of this lesson, you will be able to understand the basic functionality of COSMOSFloWorks and perform hydraulic analysis to the following part.
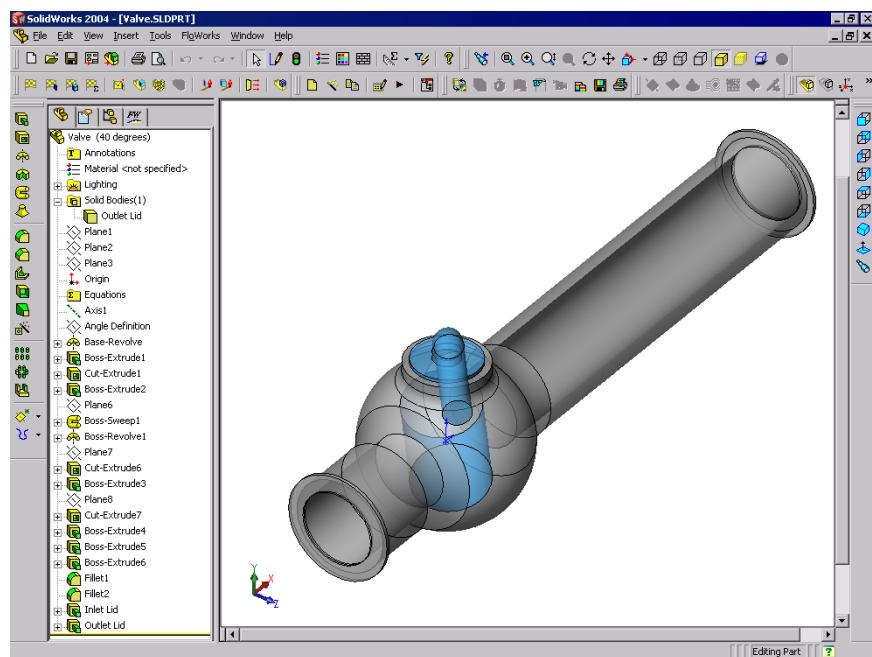
Active Learning Exercise — Determination of Hydraulic Loss

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

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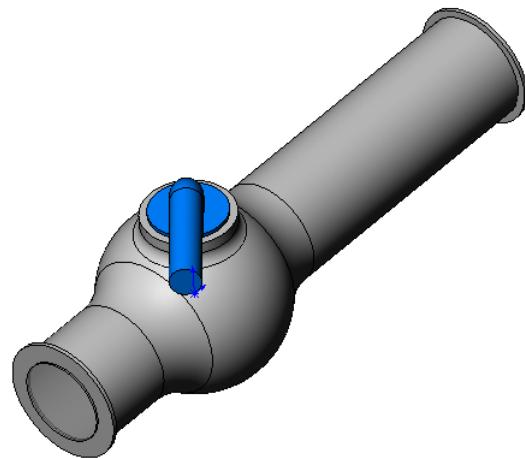
Opening the Valve.SLDPRT Document

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

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

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

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

Checking the COSMOSFloWorks Menu

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

File   Edit   View   Insert   Tools   FloWorks   Window   Help

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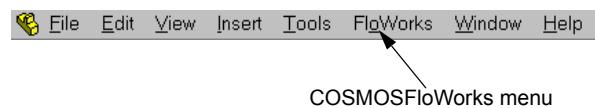
**1** Click **Tools**, **Add-Ins**.

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

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

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

Model Description

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

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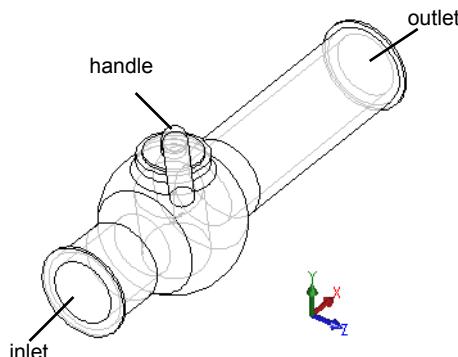
The standard engineering definition of a hydraulic resistance of an obstacle in a pipe is the difference between the total pressures (i.e. where a stream is not disturbed by the obstacle) upstream and downstream of the obstacle (the valve in our case) divided by the incoming dynamic head, from which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

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

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

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

Creating Lids

Creating Inlet Lid

1 Select the face shown in the picture.

2 Click **Sketch** on the Sketch toolbar.
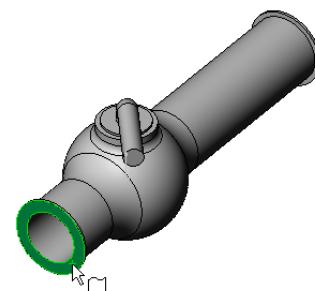
3 Select the tube's edge.

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

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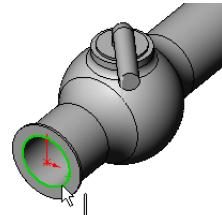
6 Click **Extrude Boss/Base** on the Features toolbar.

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

- End Condition = **Mid Plane**

- Depth = 0.005

8 Click **OK** to create the inlet lid.

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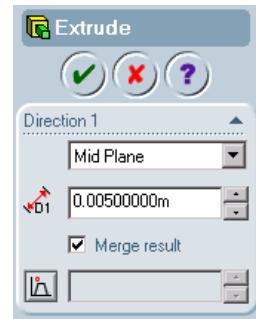
Creating Outlet Lid

9 Select the face shown in the picture.

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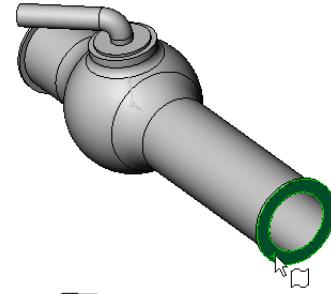
11 Select the tube's edge.

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

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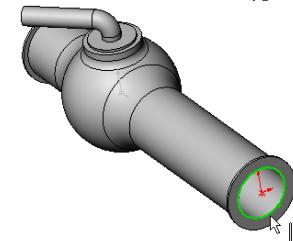
Not sure you have created the lids properly? COSMOSFloWorks can easily check your model for possible geometry problems.

Checking the Geometry

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

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

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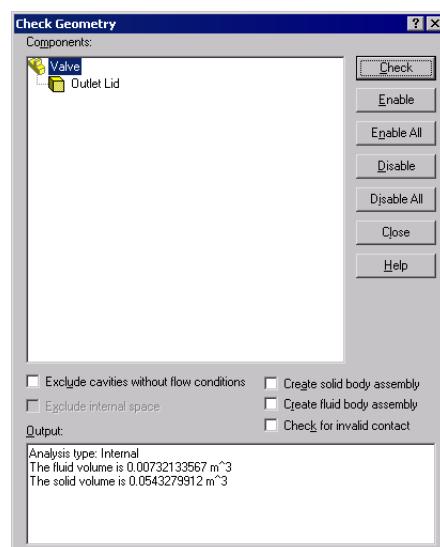
The first step in performing flow analysis is to create a COSMOSFloWorks project.

Creating a Project

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

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

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

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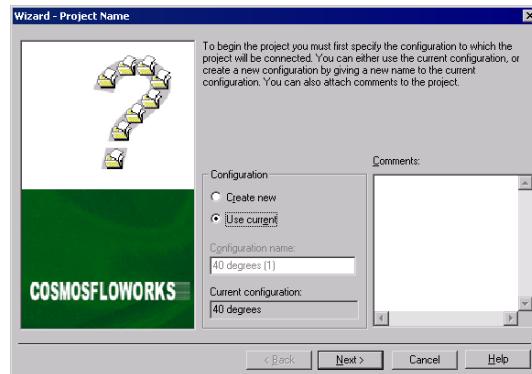
3 In the **Units** dialog box you can select the desired system of units for both input and output (results).

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

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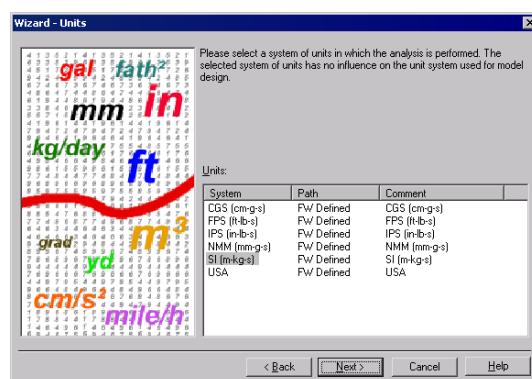
4 In the **Fluid Type and Physical Features** dialog box you can select the fluid type.

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

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

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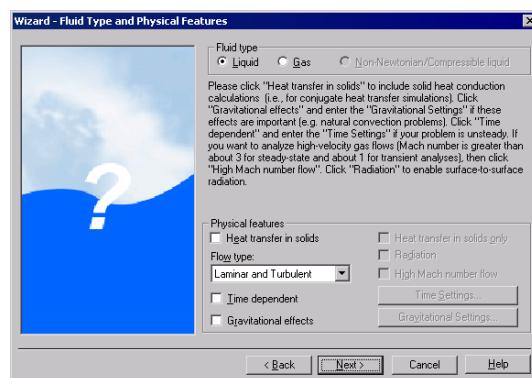
5 In the **Analysis Type** dialog box you can select either **Internal** or **External** type of the flow analysis.

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

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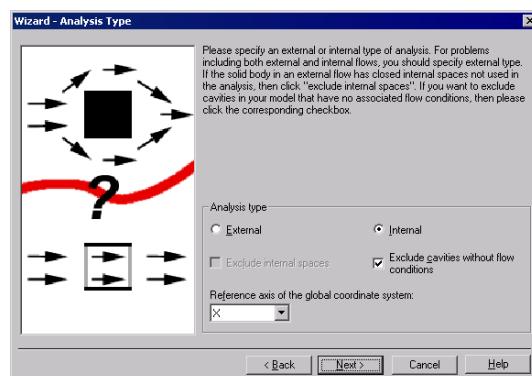
6 In the **Roughness** dialog box you can specify the wall roughness value applied by default to all model walls. To specify the roughness value for a specific wall you must create a **Real Wall** boundary condition.

In this project we will not concern rough walls.

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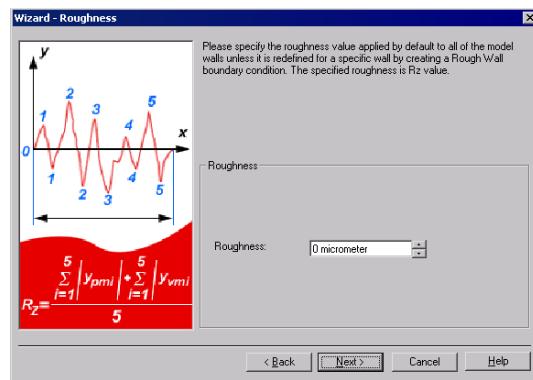
7 In the **Selecting Fluid Substances** dialog box you can select the fluid substances whose flow is analyzed in the COSMOSFloWorks project.

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

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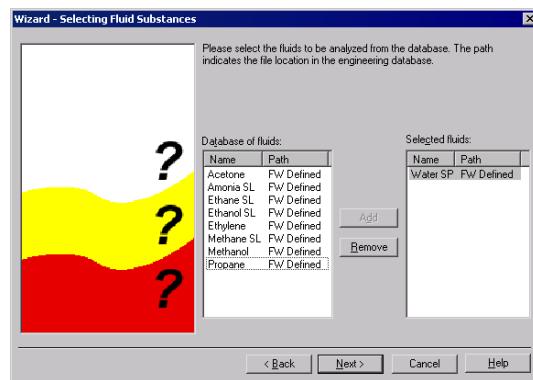
Note: Engineering Database contains numerical physical information on a wide variety of gas, liquid and solid substances as well as radiative surfaces. You can also use the Engineering Database to specify a porous medium. The Engineering Database contains pre-defined unit systems. It also contains fan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

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

---

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

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

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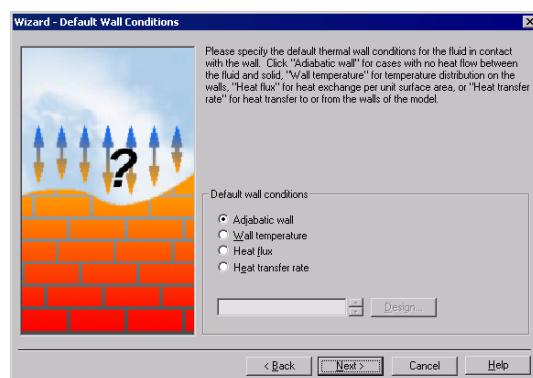
9 In the **Initial and Ambient Conditions** dialog box specify initial values of the flow parameters. For steady internal problems these values specified closer to the expected flow field will reduce the analysis time.

For this project use the default values.

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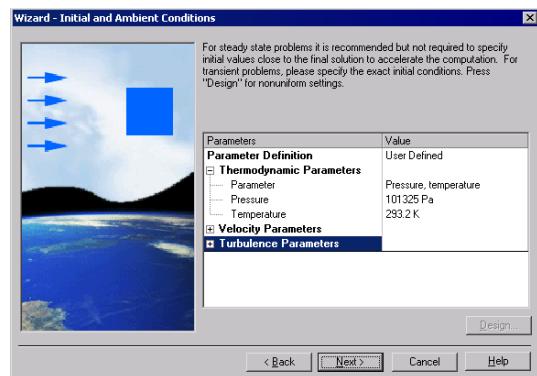
**Note:** For steady flow problems COSMOSFloWorks iterates until the solution convergences. For unsteady (transient, or time-dependent) problems COSMOSFloWorks marches in time for a period you specify.

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

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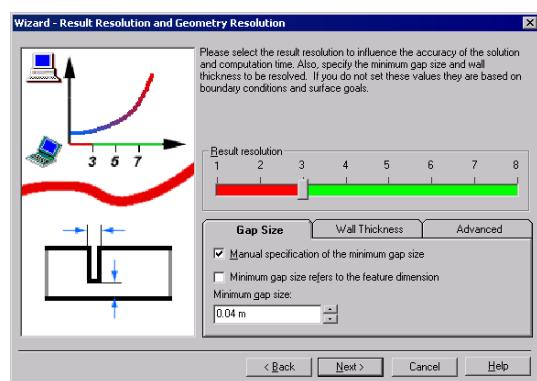
Result Resolution governs the solution accuracy that can be interpreted as resolution of calculation results. You specify result resolution in accordance with the desired solution accuracy, available CPU time and computer memory. Because this setting has an influence on the number of generated mesh cells,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

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

Select the **Manual specification of the minimum gap size** check box and enter `0.04 m` for the minimum flow passage.

> **Note:** COSMOSFloWorks calculates the default minimum gap size and minimum wall thickness using information about the overall model dimensions, the computational domain, and faces on which you specify conditions and goals. However, this information may be insufficient to recognize relatively small gaps and thin model walls. This may cause inaccurate results. In these cases, the Minimum gap size and Minimum wall thickness must be specified manually.

Click **Next**.

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

COSMOSFloWorks Design Tree

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

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

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

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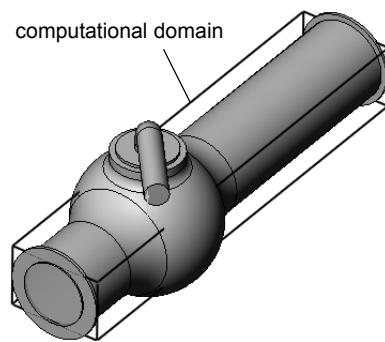
Now let us specify the other parts of the project.

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

Specifying Boundary Conditions

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

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

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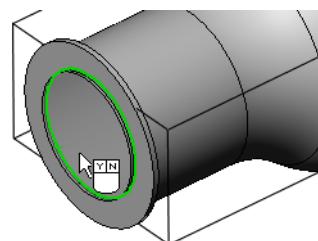
3 In the **Type of boundary condition** list, select the **Inlet Velocity** item, and then click the **Settings** tab.
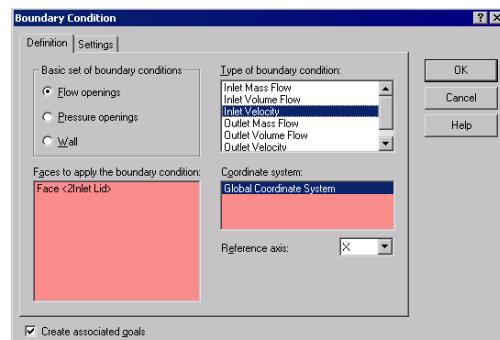
4 Double-click the **Value** cell of the **Velocity normal to face** item and set it equal to 1 (type the value, the units appears automatically).

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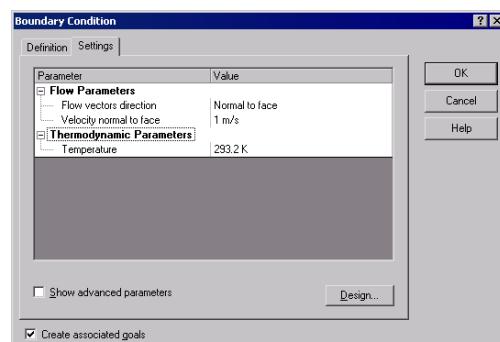
By specifying this condition we defined at the ball valve pipe inlet the water enters the valve with the velocity of 1.0 m/s.

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

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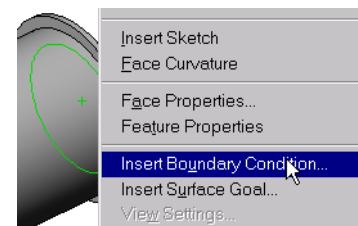
Let us specify pressure on this boundary, otherwise the problem specification is deficient. Before the calculation starts, COSMOSFloWorks checks the specified boundary conditions for mass flow rate balance. The specification of boundary conditions is incorrect if the total mass flow rate on the inlets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

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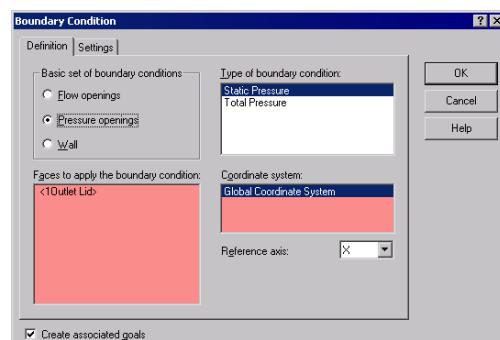
7 Accept the default values for static pressure (101325 Pa), temperature (293.2 K) and all the other parameters.
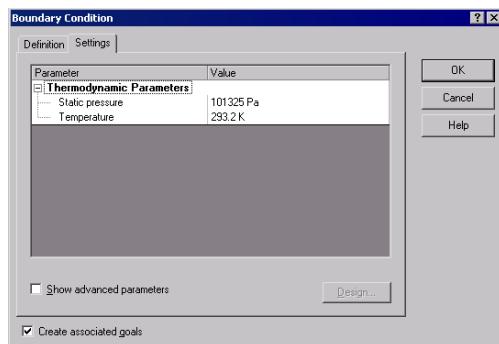
8 Click **OK**.

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

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

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

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

Specifying Surface Goals

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

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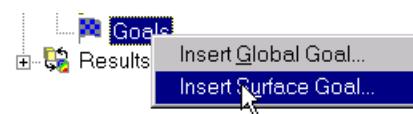
To easily select a face, simply click the **Inlet Velocity1** item in the COSMOSFloWorks design tree. The face belonging to the specified boundary condition is automatically selected and appears in the **Faces to apply the surface goal** list.

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

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

5 Click **OK**.

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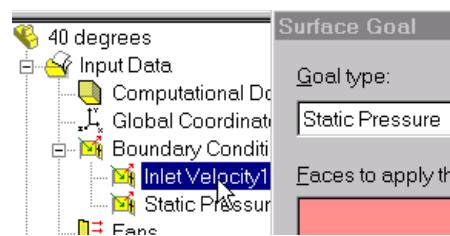
Note: Another way to rename an item is to right-click the item and select **Properties**.

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

8 Click the **Static Pressure1** item to select the inner face of the **Outlet Lid.**
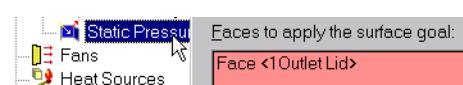
9 In the **Goal type** list, select **Total Pressure**.

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

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

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

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

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

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

16 Click-pause-click the new **SG Average Dynamic Pressure1** item and rename it to SG Average Dynamic Pressure Inlet.
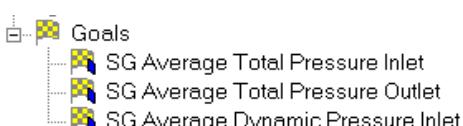
The value of the dynamic pressure at the inlet can be calculated manually. We specified the dynamic pressure goal just for convenience in the further calculation for hydraulic losses.

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

Specifying the Equation Goal

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

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

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

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

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

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

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

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

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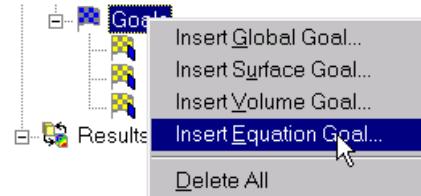
Note: To set an Equation Goal you can use only specified goals (including previously specified Equation Goals) and constants. If constants signify some physical parameters (i.e. length, area etc.) make sure of using the project's system of units. COSMOSFloWorks has no information about the physical meaning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

---

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

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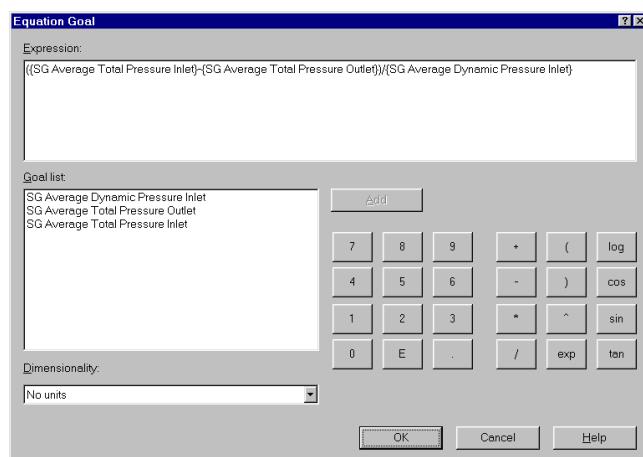
Now the COSMOSFloWorks project is ready for the calculation. COSMOSFloWorks will finish the calculation when the steady-state average value of total pressure calculated at the valve inlet and outlet are reached.

Running the Calculation

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

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

The calculation should take about 11 minutes to run on a 1.7 GHz PIV computer.

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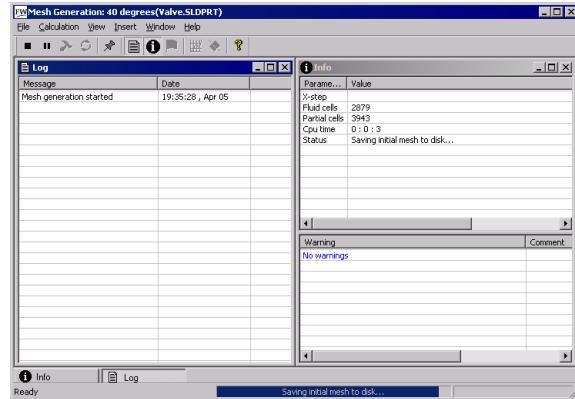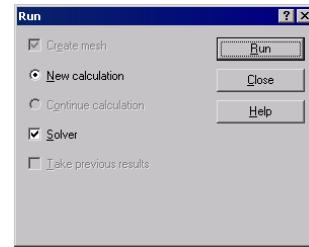
Monitoring the Solver

This is the solution monitor dialog box. On the left is a log of each step taken in the solution process. On the right is an information dialog box with mesh information and any warnings concerning the analysis.

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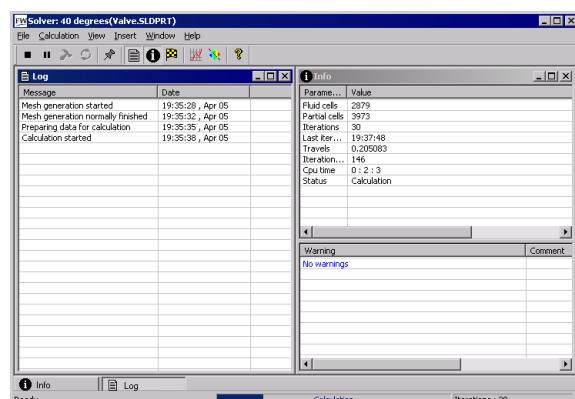
Creating Goal Plot

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

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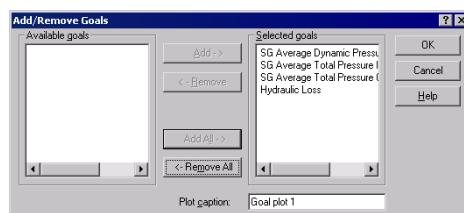
This is the Goals dialog box and each goal created earlier is listed above. Here you can see the current value and graph for each goal as well as the current progress towards completion given as a percentage. The progress value is only estimate and the rate of progress generally increases with time.
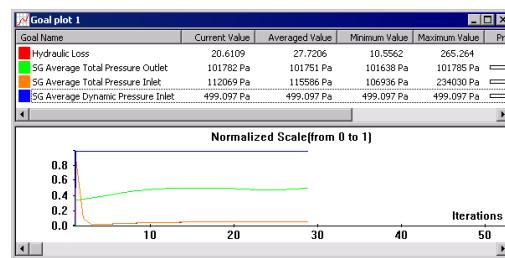
Preview Results

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

2 Click the FeatureManager tab .
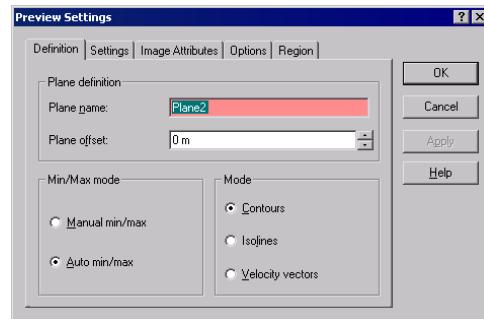
3 Select Plane 2.

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

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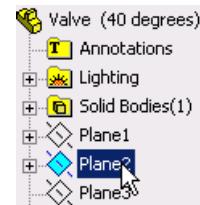
> **Note:** You can specify a parameter you want to display in the preview plane, the parameter range and display options for velocity vectors at the **Setting** tab of the **Preview Settings** dialog box.

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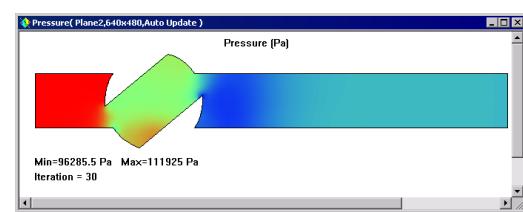
At the start of the run the results might look odd or change abruptly. However, as the run progresses these changes will lessen and the results will settle in on a converged solution. The result can be displayed either in contour-, isoline- or vector-representation.

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

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

Accessing the Results

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

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

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

Creating a Cut Plot

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

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

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

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

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

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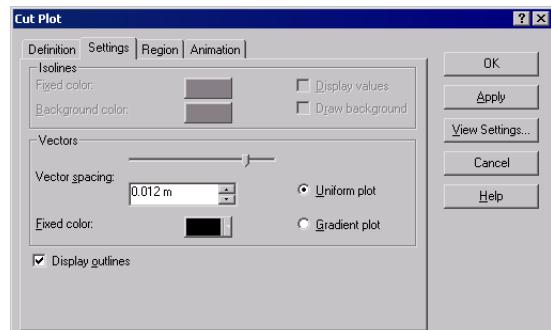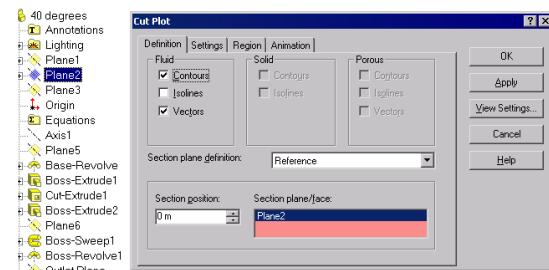
Note: The settings made in the **View Settings** dialog box refer to all **Cut Plots**,
Surface Plots, **Isosurfaces**, and **Flow Trajectories** specific features.
These settings are only applied for the active pane of the SolidWorks
graphics area. For example, the contours in all cut and surface plots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

---

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

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

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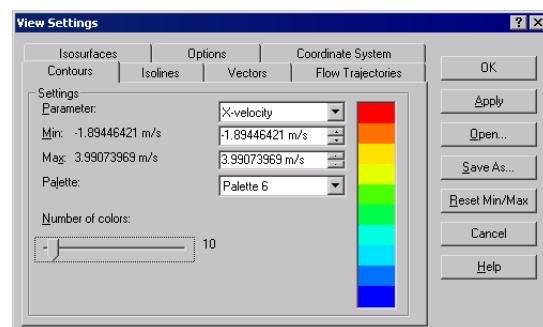
However, the cut plot is not seen through the model. In order to see the plot, you can hide the model by clicking **FloWorks**, **Results**, **Display**, **Geometry**.
Alternatively, you can use the standard SolidWorks **Section View** option.

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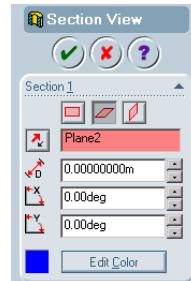
10 In the COSMOSFloWorks design tree, right-click the **Computational Domain** icon and select **Hide**.

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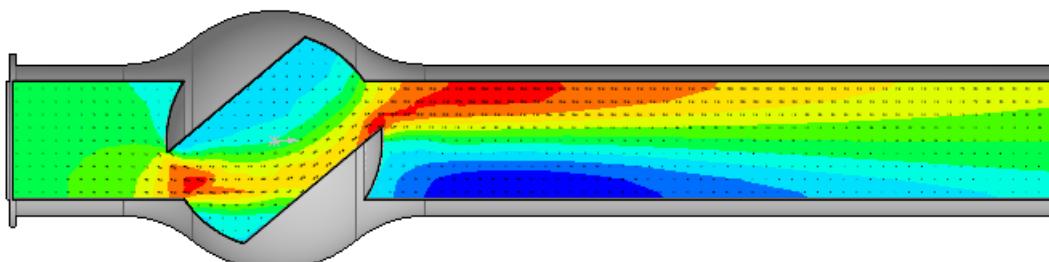
For better visualization of the vortex you can scale small vectors:

11 In the COSMOSFloWorks design tree, right-click the **Results** icon and select **View Settings**.
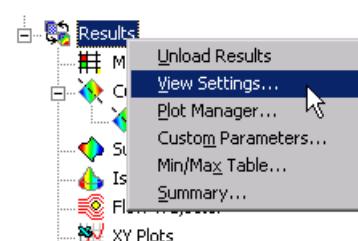
12 In the **View Settings** dialog, click the **Vectors** tab and in the **Arrow size** box type 0.02.

13 Change the **Min** value to `2 m/s`.

By specifying the custom **Min** we change the vector length so the vectors whose velocity is less than the specified Min value will have the same length as the
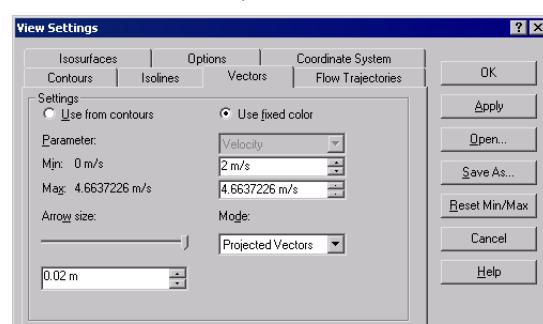
vectors whose velocity is equal to the Min. This allows us to visualize the low velocity area in more details.

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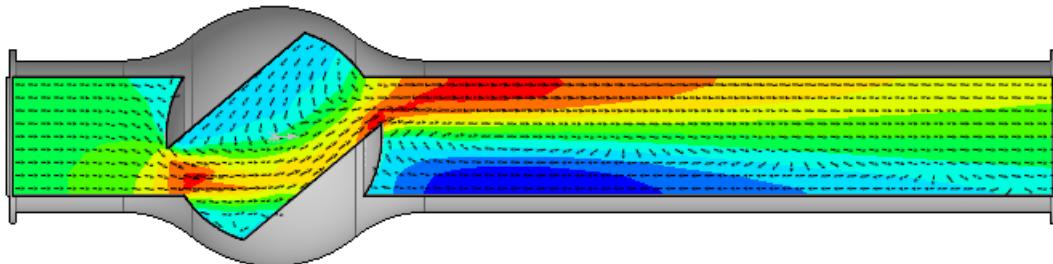
Displaying Flow Trajectories

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

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

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

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

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

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

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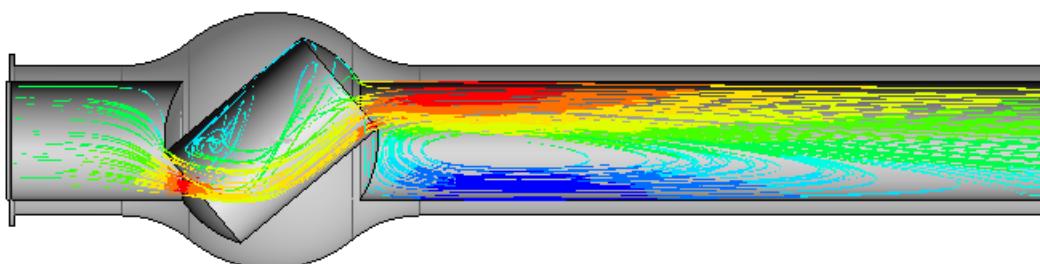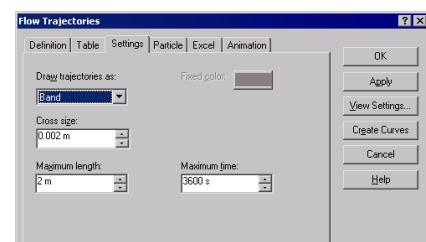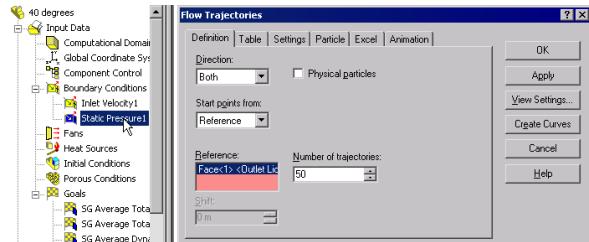
Rotate the model to better view the 3D structure of the vortices.

Creating a Goal Plot

The Goal Plot allows you to study goal changes in the course of the calculation. COSMOSFloWorks uses Microsoft Excel to display 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 COSMOSFloWorks design tree, under **Results**, right-click the **Goals** icon and select **Create**. The **Goals** dialog box appears.

2 Click **Add All**.

3 Click **OK**. The goals1 Excel workbook is created.
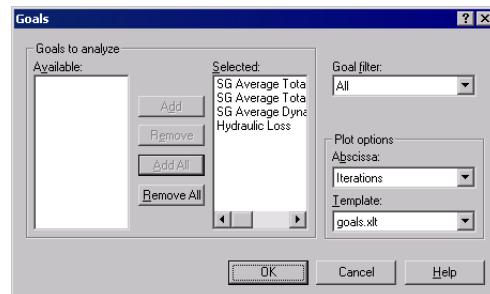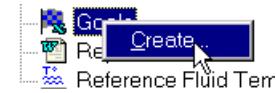
This workbook displays how the goal has being changed during the calculation. You can take the total pressure value presented in the **Summary** sheet.

Goal Name	Unit	Value	Averaged Value	Minimum Value	Maximum Value	Progress [%]	Use In Convergence
SG Average Total Pressure Inlet	[Pa]	111892.897	111853	111773	111902	100	Yes
SG Average Total Pressure Outlet	[Pa]	101989.3353	101933	101815	101993	100	Yes
SG Average Dynamic Pressure Inlet	[Pa]	499.09671	499.097	499.097	499.097	100	Yes
Hydraulic Loss	[ ]	19.84297122	19.8758	19.8182	20.0334	0	No

Cloning Project

The calculation performed yields the total hydraulic resistance ξ including both valve's hydraulic resistance ξv (due to the obstacle) and the tubes' hydraulic resistance due to friction ξ_f: $\xi = \xi v + \xi_f$. 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°.

You can create a new COSMOSFloWorks project in three ways:

• The Project Wizard is the most straightforward way of creating a COSMOSFloWorks 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 COSMOSFloWorks project by using a Template, either a default template or custom template created from a previous COSMOSFloWorks 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° angle and specify the same condition as the 40° angle project is to clone the existing **40 project**.

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

2 Click **Crete New**.

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

4 Click **OK**.
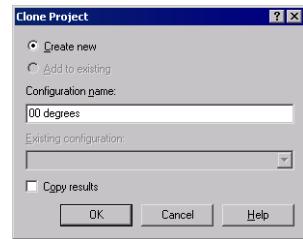
Now the new COSMOSFloWorks project is attached to the new **00 degrees** configuration and it has inherited all the settings from the **40 degrees** project. All our input data are copied, so we do not need to define them again. All changes will only be applied to this new configuration, so the old results are still saved.

Changing the Valve Angle

1 In the FeatureManager, right-click the **Angle Definition** plane and select **Edit Definition**.

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

3 Click **OK** .
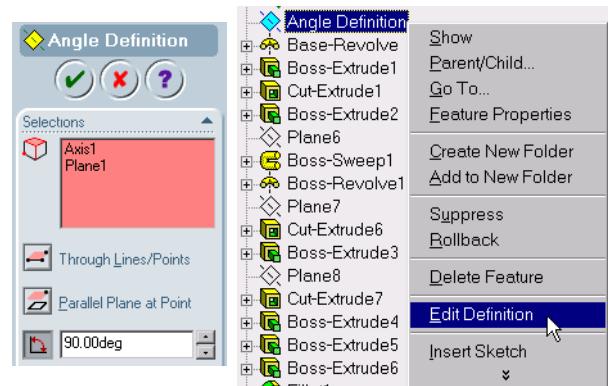
After clicking OK, two warning messages appear asking you to rebuild the computational mesh and to reset the computational domain.
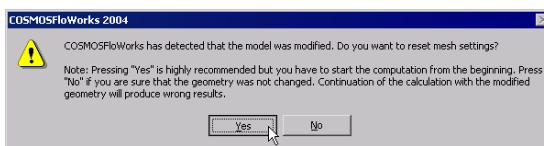
4 Select **Yes** for both the messages.

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 and, in turn, more computer time and memory will be required. To solve your task in the most effective way you should choose the optimal settings for the task.

1 Click **FloWorks**, **Initial Mesh**.

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

2 Click **OK**.
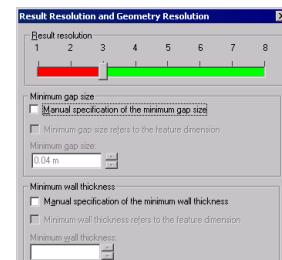
Changing the Computational Domain

You can take advantage of the symmetry of the straight pipe to reduce the CPU time and memory requirements for the computation. Since this 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. Symmetric geometry can produce nonsymmetric flow. In this case, the flow in the straight pipe is symmetric so we can reduce the computational domain.

---

1 In the COSMOSFloWorks 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 resulting in decreases in memory requirements and CPU time. 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.
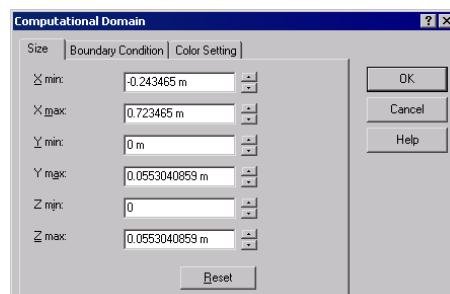
5 In the **At Y min** and **At Z min** lists select **Symmetry**.

6 Click **OK**.

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

symmetry
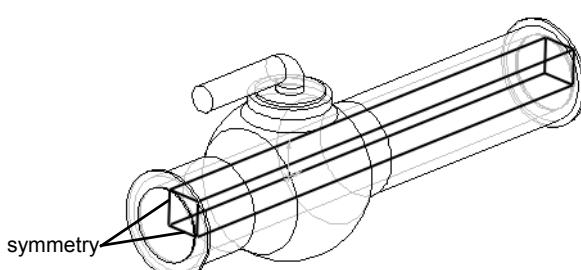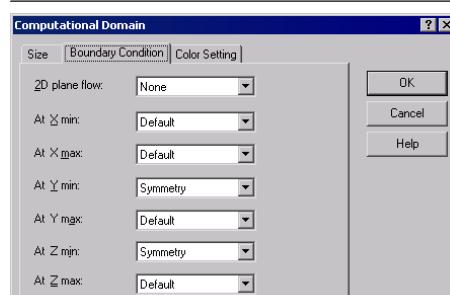
Getting the Valve's Hydraulic Loss

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

Valve.SLDPRT [00 degrees]

Goal Name	Unit	Value	Averaged Value	Minimum Value	Maximum Value	Progress [%]	Use In Convergence
SG Average Total Pressure Inlet	[Pa]	101856.8187	101864	101857	101882	100	Yes
SG Average Total Pressure Outlet	[Pa]	101810.7632	101808	101805	101811	100	Yes
SG Average Dynamic Pressure Inlet	[Pa]	499.09671	499.097	499.097	499.097	100	Yes
Hydraulic Loss	[ ]	0.092277782	0.111959	0.0922778	0.152809	0	No

Now you can calculate the valve's hydraulic loss in the ball valve whose handle is turned by 40°.

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 COSMOSFloWorks 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 COSMOSFloWorks. Once the oscillations are less than the convergence criterion the goal becomes converged.

To determine the parameter's steady-state value more accurately, it would be more correct to use the value averaged over the analysis interval is shown in the **Averaged Value** column.

Total hydraulic losses (40 deg)	Friction losses (0 deg)	Valve's loss
19,87	0,11	19,76

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

1 What is COSMOSFloWorks?

2 How do you start a COSMOSFloWorks session?

3 What is a fluid flow analysis?

4 Why analysis is important?

5 What kind of analyses is typical for COSMOSFloWorks internal flow analyses?

6 What is the specific requirement of COSMOSFloWorks internal analyses?

7 How can you ensure the model is closed?

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

9 What is the first step to start a COSMOSFloWorks analysis?

10 In what ways can a COSMOSFloWorks project be created?

11 How do you specify a fluid for a project?

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

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

_____

_____

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

_____

_____

_____

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

_____

16 How do you start a calculation?

_____

17 What needs to be done first before viewing the result information?

_____

18 What display options are available in COSMOSFloWorks to view the calculation results?

_____

_____

_____

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

_____

_____

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

_____

_____

Project — 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 COSMOSFloWorks 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 **temp\Education Examples\Bilateral Sudden Expansion** 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 to see 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**.
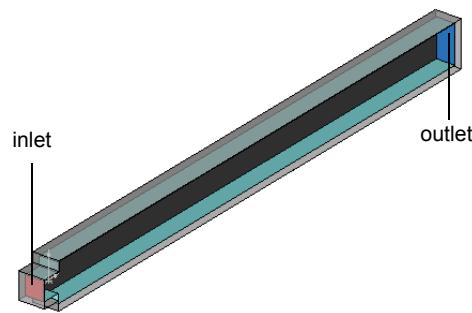
2 Using the **Wizard**, create the COSMOSFloWorks project for internal water analysis with the **Result resolution** level set to 5 (all other settings are default).

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?

Answer: _____

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?
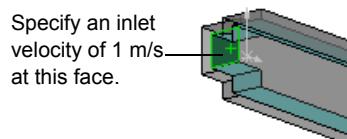
Answer: _____

5 Specify 2D X-Y plane flow analysis.

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.
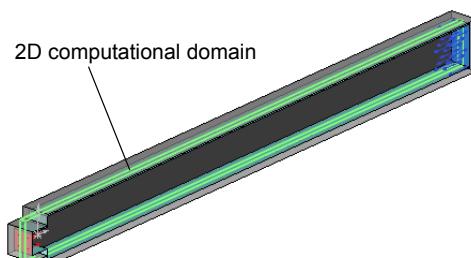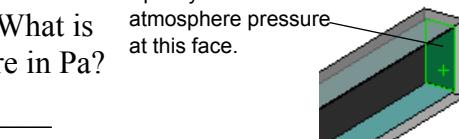
To consider the hydraulic resistance due to the sudden expansion only, let us replace in the calculations the channel's real walls by the "Ideal Walls" boundary condition option in COSMOSFloWorks, which apply adiabatic frictionless walls. As a result, any wall friction will be absent (of course, this can be done in calculations only and it is impossible in physical experiments). As for 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 (colored cyan).

Make these walls ideal.

7 Specify the Total Pressure and Dynamic Pressure surface goals at inlet.

8 Specify the Total Pressure surface goal at outlet.

9 Specify the Equation goal calculating the total hydraulic loss.

Make these walls ideal.

10 Run the calculation.

11 Plot the velocity distribution along the channel.

12 Obtain the hydraulic loss caused by the model's sudden expansion by viewing the equation goal.
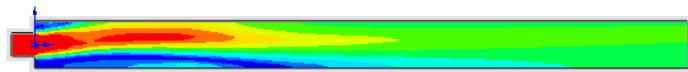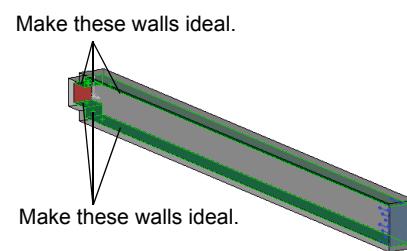
Answer: _____

Lesson 1 Vocabulary Worksheet

Name_____Class: _____ Date:_____

Fill in the blanks with the proper words.

1 The fluid flow equations solved by COSMOSFloWorks: _____

_____

2 The method used for solving these equations with COSMOSFloWorks: _____

_____

3 The method used for solving time-independent problems with COSMOSFloWorks: ___

_____

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

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

6 The feature that allows users to track the convergence of a flow parameter(s) in a COSMOSFloWorks project: _____

7 The physical feature which must be selected in COSMOSFloWorks to initiate temperature calculation in solids: _____

8 The physical feature which must be selected in COSMOSFloWorks to obtain a time-dependent solution: _____

9 The physical feature which must be selected in COSMOSFloWorks to calculate a flow with significant supersonic regions: _____

10 The physical feature which must be selected in COSMOSFloWorks to properly calculate a heat convection and/or mixing fluids in low-velocity flows not in weightlessness: _____

11 The physical feature which must be selected in COSMOSFloWorks to fully suppress any flow turbulence in the computational domain: _____
The COSMOSFloWorks approach of specifying a distributed resistance to a fluid flow:

_____

12 Liquids whose viscosity depends on flow velocity gradients: _____

Lesson 1 Quiz

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 COSMOSFloWorks for an internal analyses?

_____

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

_____

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

_____

_____

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

_____

_____

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

_____

_____

_____

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

_____

7 Can you obtain intermediate results during the calculation?

_____

8 How do you load results?

_____

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

_____

_____

_____

10 When can the symmetry condition be applied?

_____

11 What causes hydraulic losses in a pipeline?

_____

_____