1

First Steps - Ball Valve Design

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

Conventions

- **Bold** refers to buttons, icons or tabs to click on.
- **Courier** refers to information you need to enter using the keyboard.
- **Menu, Submenu** indicates menu picks found in the main menu.
- Italic is used for helpful notes and explanations.

Open the SolidWorks Model

- 1 Click File, Open. In the Open dialog box, browse to the Ball Valve.SLDASM assembly located in the First Steps - Ball Valve folder and click Open (or double-click the assembly). Alternatively, you can drag and drop the Ball Valve.SLDASM file to an empty area of SolidWorks window. Make sure, that the default configuration is the active one.
- This is a ball valve. Turning the handle closes or opens the valve. The mate angle controls the opening angle.
- 2 Show the **lids** by clicking the features in the SolidWorks Feature Manager (Lid 1 and Lid 2).



- We utilize this model for the COSMOSFloWorks simulation without many significant changes. The user simply closes the interior volume using extrusions we call lids. In this example the lids are semi-transparent allowing a view into the valve.
- **3** On the **Visualisation** tab, set the **Background** color to white.

Create a COSMOSFIoWorks Project

- 1 Click FloWorks, Project, Wizard.
- 2 Once inside the Wizard, select **Create new** in order to create a new configuration and name it Project 1.
- COSMOSFloWorks will create a new configuration and store all data in a new folder.



Click Next.

Create a COSMOSFIoWorks Project

- Choose the system of units (SI for this project). Please keep in mind that after finishing the Wizard, you can change the unit system anytime with FloWorks, Units.
- Construction
 Construction

 Phase select a system of units in which the analysis is performed. The electric system of units has no influence on the unit system used for model design.
 Particle System of units has no influence on the unit system used for model design.

 White
 Units:
 Descent system of units in which the analysis is performed. The design.

 gradifying
 Path
 Comment

 GCSS (omgray)
 PWD befined
 CSS (omgray)

 PTS (hits)
 PVD befined
 PSS (hits)

 IPS (hits)
 PVD befined
 PSS (hits)

 IPS (hits)
 PVD befined
 PSS (hits)

 ISI(mix)
 PVD befined
 VDA(reg)

 USA
 PVD befined
 VDA(reg)
- Within COSMOSFloWorks, there are some predefined systems of units. You can also define your own and switch between them anytime.

Click Next.

4 Set the fluid type to **Liquid**. Do not include any other physical features.

	Fluid type If Liquid C Gas	C Montkextonian@ampressible.iau/d
	Please click "Heat transfer in sc celouelistics, "in for conjugate "Gravitational effects" and enter are important (e.g. natural conv and enter the "Time Stetings" (i analyze high-velocity gas fows steady-state and about 1 for the number flow". Click "Padatation" Click "Host treader in solitis on	Sids" to include solid heat conduction heat transfer simulationa). Click the "Gravitational Settings" if these effects ection problems). Click "Time dependent" your problem is unstaedy. If you want to (Mach number is greater than about 3 for nient analyses), then click. "High Mach to enable surface-to-surface radiation. K" in a fut invisits in your heat teredar.
	analysis. Under "Flow type", yo Laminar Only or Turbulent Only	u can choose to consider the flow as
	endysis. Under "Flow type", yo Laminar Only or Turbulent Only Physical features If Hgat transfer in solids	u can choose to consider the flow as
	enalysis, Under "Flow type", yo Leminer Only or Turbulent Only Physical features Flog type:	 Conclusion to consider the flow as Heattransfer in sorids gridy Regulation
~	analysis Under "Flow type", yo Laminar Only or Turbulent Only Physical loatures IF Heat transfer in solids Flow type: Laminar and Turbulent	U con choose to consider the flow as I Heat transfer in sol ds gnly I Registron I Figh Machinember flow
~	endysis Under "Flow type", to Lammer Only or Turbulent Only Physical features Heat transfer in solids Flow type: Laminar and Turbulent	Con choose to consider the flow as Heat transfer in social gnly Pagheton Figh Mach number flow Time Settings

COSMOSFloWorks can analyze either

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

Click Next.

Create a COSMOSFIoWorks Project

COSMOSFIoWorks 2004 Tutorial

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

Click Next.

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





- 7 Choose Water SP as the fluid. You can either double-click Water SP or select the item in the left column and click Add.
- COSMOSFloWorks has an integrated database containing several liquids, gases and solids. Solids are used for conduction in conjugate heat transfer analyses. You can easily create your own materials. Up to ten liquids or gases can be chosen for each analysis run.

Please select the fluids to be analyzed from the database. The path indicates the file flootion in the engineering database. Pathone St. PVD B Propose of fluids: Name Path Acations St. PVD B Band St. PVD B Band St. PVD B Methods St. PVD B Band St. PVD B Methods St. PVD B Band St. PVD B Methods St. PVD B Methods St. PVD B Methods PVD B Methods St. EvD B Methods PVD B Methods St. EvD B Methods

Click Next.

Create a COSMOSFIoWorks Project

- 8 Click Next accepting the default wall conditions.
- *Since we did not choose to calculate* the amount of heat transfer within the solids we have the option of defining a value of heat transfer for the surfaces in contact with the fluid. This box is where we can set the default wall type. Leave the default Adiabatic wall specifying the walls are perfectly insulated (no heat escapes through the walls).
- 9 Click **Next** accepting the default for the initial conditions.
- This box is where we can change the default settings for pressure, *temperature and velocity. The closer* these values are set to the final values determined in the analysis, the quicker the analysis will finish. Since we do not have any knowledge of the expected final values, we will not modify them for this demonstration.

10 Accept the default for the **Result Resolution**.

Result Resolution is a measure of the desired level of accuracy of the results. It controls not only the resolution of the mesh, but also sets many parameters for the solver, e.g. the convergence criteria. The higher the Result Resolution, the finer the mesh will be and the stricter the



-	For steady state problems it is recomm initial values close to the final solution to transient problems, please specify the i "Design" for nonuniform settings.	ended but not required to specify accelerate the computation. Fo exact initial conditions, Press
	Parameters	Value
	Parameter Definition Thermodynamic Parameters Parameter Pressure Temperature Velocity Parameters G Turbulence Parameters	User Defined Pressure, temperature 101325 Pa. 2932 K.



convergence criteria will be set. Thus, Result Resolution determines the balance between precise results and computation time. Entering values for the minimum gap size and minimum wall thickness is important when you have small features.



Create a COSMOSFIoWorks Project

COSMOSFIoWorks 2004 Tutorial

Setting these values accurately ensures your small features are not "passed over" by the mesh. For our model we type the value of the minimum flow passage as the minimum gap size.

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

Click Next.

- 11 Click Finish.
- This box provides an overview of the selections made during the wizard.



Now COSMOSFloWorks creates a new configuration with the COSMOSFloWorks data attached.

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

- Notice the name of the new configuration has the name you entered in the Wizard.

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

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



🖃 🙀 Results

Boundary Conditions

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

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



Boundary Conditions

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

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



Boundary Conditions

COSMOSFIoWorks 2004 Tutorial

3 Select Flow openings and Inlet Mass Flow.



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



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



- With the definition just made, we told COSMOSFloWorks that at this opening 0.5 kilogram of water per second is flowing into the valve. Within this dialog box we can also specify a swirl to the flow, a non-uniform profile and time dependent properties to the flow. The mass flow at the outlet does not need to be specified due to the conservation of mass; mass flow in equals mass flow out. Therefore another different condition must be specified. An outlet pressure should be used to identify this condition.
- 6 Select the **inner** face of the **Lid 2** part as shown. (To access the inner face, right-click the mouse to cycle through the faces under the cursor until the inner face is highlighted, then click the left mouse button).
- 7 Right-click the **Boundary Conditions** icon and select **Insert Boundary Condition**.



Boundary Conditions

8 Select Pressure openings and Static Pressure.



9 Keep the defaults under **Settings**.



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



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

Define the Engineering Goal

1 Right-click the COSMOSFloWorks Design Tree Goals icon and select Insert Surface Goal.

- 2 Click the **Inlet Mass Flow1** item to select the face where it is going to be applied
- **3** Select **Static Pressure** as **Goal type**.
- 4 Accept to **Use the goal for convergence control** and keep the **Average Value**.
- □ If the Use the goal for convergence control check box is not selected for a goal, it will not influence the task stopping criteria. Such goals can be used as monitoring parameters to give you additional information about processes occurring in your model without affecting the other results and the total calculation time.
- 5 Click OK. The new SG Average Static Pressure1 item appears in the COSMOSFloWorks Design Tree.

E-Stat	ic Pressure1 ources conditions Conditions sert <u>G</u> lobal Goal sert Surface Goal sert ⊻olume Goal
Computational Domain Global Coordinate System Coordinate System Component Control Component Control Com	Surface Gool Gool type: Static Prossure Eaces to apply the surface goal Face <1Lid1@1>
Surface Goal	1×
Goal type:	Galculate:
Face <1Lid-2@1>	¹ ылылда дайа — Т
Use the goal for convergence contro Coordinate system	l



Engineering goals are the parameters in which the user is interested. Setting goals is in essence a way of conveying to COSMOSFloWorks what you are trying to get out of the analysis, as well as a means of reducing the time COSMOSFloWorks takes to reach a solution. By only selecting the variable which the user desires accurate values for, COSMOSFloWorks knows which variables are important to converge upon (the variables selected as goals) and which can be less accurate (the variables not selected as goals) in the interest of time. Goals can be set throughout the entire domain (Global Goals), in a selected area (Surface Goal) or within a selected volume (Volume Goal). Furthermore, COSMOSFloWorks can

consider the average value, the minimum value or the maximum value for goal settings. You can also define an Equation Goal that is a goal defined by an equation (basic mathematical functions) with the existing goals as variables. The equation goal allows you to calculate the parameter of interest (i.e., pressure drop) and keeps this information in the project for later reference.

Click File, Save.

Solution

- 1 Click FloWorks, Solve, Run.
- 2 Click Run.
- The solver should take a few minutes to run on a 500MHz PIII platform.

	Bun
New calculation	Close
	Help
Solver	
Teke prevous results	

Monitor the Solver

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

Log		X Info	
Message	Date	Parameter	Value
Mesh generation started	14:02:39 . Jul 24	Fluid cells	1712
vesh generation normally fi	14:02:54 , Jul 24	Partial cells	2017
Preparing data for calculation	14.03:30 Jul 24	Iterations	9
Calculation started	14:04:30, Jul 24	Last derati	14.05:37
		Travels	0.11564
		Iterations	77
		Cputime	0:1:7
		Status	Calculation
		Warning	
		No warnings	· ·

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

Monitor the Solver

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



- **3** Click **Insert Preview (b)** on the **Solver** toolbar.
- 4 This is the Preview Settings dialog box. Selecting any SolidWorks plane from the SolidWorks Feature Manager Tree and pressing OK will create a preview plot of the solution on that plane. For this model Plane2 is a good choice to use as the preview plane. The preview plane can be chosen anytime from the SolidWorks Feature Manager.
- The preview allows one to look at the results while the calculation is still running. This helps to determine if all the boundary conditions are correctly defined and gives the user an idea of how the solution will look even at this early stage. At the start of the run





the results might look odd or change abruptly. However, as the run progresses these changes will lessen and the results will settle in on a converged solution. The result can be displayed either in contour-, isoline- or vector-representation.

🗄 뎛 Results

🔶 Surfac

👍 Isosurfaces

Insert

🗄 🗟 Results Load Results Select Results Plot Manager... Custo<u>m</u> Parameters

Access the Results

5

COSMOSFIoWorks 2004 Tutorial

- 1 Click FloWorks, Results, Display, Transparency and set the model transparency to 0.75.
- The first step for results is to generate a transparent view of the geometry, a 'glass-body'. This way, you can easily see where cut planes etc. are located with respect to the geometry.
- 2 Right-click the **Results** icon and select Load **Results** to activate the postprocessor.
- 3 In the Load Results dialog box, keep the default project's results file (1.fld) and click **Open**.

Cut	Dinte
Guι	LIO12

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

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

Access the Results







Apply

Help

w Settings. Cancel

F Conto

r legines

E Vectors

Cut Plots

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

This is the plot you should see.

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

Cut Plo

Fluid

Contours

□ [solines

☐ Vectors

Section plane defin

Definition Settings Region Animation

-

Solid

Contours

E legines

Reference

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



Change the contour cut plot to a vector cut plot. To do this, right-click the Cut Plot 1 icon and select Edit Definition.





Surface Plots

- 6 Clear Contours and select Vectors in the plot definition.
- 7 Click OK.



This is the plot you should see.

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



Surface Plots

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

- 1 Right-click the Surface Plots icon and select Insert.
- 2 Select the Use all faces check box.
- 3 Select the **Contours** check box.
- The same basic options are available for Surface Plots as for Cut Plots. Feel free to experiment with different combinations on your own.



Sylfaces -	₩ Contours	⊻iew Settings
	₩ Contours	1
	Isolines	Cancel
	T Vectors	Help
Use all faces		

Mediu

Isosurface Plots

COSMOSFIoWorks 2004 Tutorial

- 4 Click **OK** and you get the following picture:
- This plot shows the pressure distribution on all faces of the valve in contact with the fluid. You can also select one or more single surfaces for this plot, which do not have to be planar.



Isosurface Plots

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

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

This is the plot that will appear.

- The Isosurface is a 3-Dimensional surface created by COSMOSFloWorks at a constant value for a specific variable. The value and variable can be altered in the **View Settings** dialog box under the **Isosurfaces** tab.
- 2 Right-click in the white area and select **View Settings** to enter the dialog.
- **3** Go to **Isosurfaces** tab.
- 4 Examine the options under this dialog box. Try making two changes. The first is to click in the **Use from contours** so that the color of the isosurface be colored in the same manner as the pressure value on a contour plot.





Flow Trajectory Plots

- 5 Secondly, click at a second location on the slide bar and notice the addition of a second slider. These sliders can later be removed by dragging them all of the way out of the dialog box.
- 6 Click **OK** and you should see something similar to this image.



The isosurface is a useful way of determining the exact area, in 3-dimensions, where the flow reached a certain pressure, velocity or other variable.



Flow Trajectory Plots

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

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



- In the COSMOS-FloWorks Design Tree, click the Static
 Pressure1 item to select the inner face of the outlet Lid 2 part.
- 3 Set the Number of trajectories to 16.



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



For this plot we selected the outlet lid (any flat face or sketch can be selected) and therefore every trajectory crosses that selected face. The trajectories can also be colored by values of whatever variable chosen in the **View Settings** dialog box. Notice the trajectories that are entering and exiting through the exit lid. This is the reason for the warning we received during the solver. COSMOSFloWorks warns us of inappropriate analysis conditions so that we do not need to be CFD experts. When flow both enters and exits the same opening, the accuracy of the results will worsen. In a case such as this, one would typically add the next component to the model (such as a pipe extending the computational domain) so that the vortex does not occur at an opening.

XY Plots

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

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

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

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



XY Plots

Surface Parameters

COSMOSFIoWorks 2004 Tutorial

2 Choose Velocity and Pressure as Physical parameters. Select Sketch1 from the SolidWorks Feature Manager.

Leave all other options as defaults.

Click **OK**. MS Excel will open and generate two lists of data points as well as two graphs, one for Velocity and the other for Pressure. One of these plots is the one shown below. You will need to toggle between different sheets in Excel to view each graph.



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

Surface Parameters

3

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

Analyze a Design Variant in the SolidWorks Ball part

COSMOSFIoWorks 2004 Tutorial

- 1 Right-click the **Surface Parameters** icon and select **Insert**.
- 2 In the COSMOSFloWorks Design Tree, click the **Inlet Mass Flow1** item to select the **inner** face of the inlet **Lid 1** part.
- **3** Click **Apply**.
- 4 Select the **Local** tab.



- The average static pressure at the inlet face is shown to be 128724 Pa. We already know that the outlet static pressure is 101325 Pa since we applied it previously as a boundary condition. So, the average static pressure drop through the valve is calculated to be 27399 Pa.
- 5 Close the **Surface Parameters** dialog box.

Analyze a Design Variant in the SolidWorks Ball part

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

Create a new configuration using the SolidWorks Configuration Manager Tree.

Analyze a Design Variant in the SolidWorks Ball part

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

View Settings... Zoom to Selection

Hore Ball(1) (Standard) Handle Go To... Hore Clid(1) Suppress Hore Clid(2) Hide con Hore Group' Setto Lig Croup' () Ske Ex

Suppress Hide components Set to Lightweight Eix Component Properties..

Delete SmartMates Mov⊭ Component View Mates Ed: Part Ogen ball.sldgt

🔓 default

塔 Project 1

all Valve Configuration(s) (Project 1)

Add Configuration

Document Properties

4 Go to SolidWorks FeatureManager, right-click the **Ball** item and select **Open ball.sldprt**.

Create a new configuration using the SolidWorks Configuration Manager Tree.

- 1 Right-click the root item in the SolidWorks Configuration Manager and select Add Configuration.
- 2 Name the new configuration as 1,5 fillet Ball.
- 3 Click OK.



Analyze a Design Variant in the SolidWorks Ball part

COSMOSFIoWorks 2004 Tutorial

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

- 5 Back in the assembly, right-click the Ball item and select Component Properties.
- 6 At the bottom of the **Component Properties** dialog box change the configuration of the Ball part to the new filleted one.

Fillet	
Eillet Type	FU
Itoms To Fillet	
<u>Multiple radius fillet</u> Tangent propagation	[Radus: 11.5mm]
Setback Parameters 👻	
Fillet Options •	

General properties Component Name: Boll Instance Id. Full Name: Boll Model Document Name: E Vabil SLOPERT Bill Configuration specific properties Visibility If Hide Model Cglor Suppression state C Bosolved Solve as C Brooble	rowse
Component Name: Ball stance id Full Name: Ball Model Document Path: E\Dell SL DFRT. Bill Configuration specific properties Visibility IF Hide Model Cglor Suppression state C Suppression state	rowse
Model Document Path. ExBell SLOPERT. B Configuration specific properties	rowse
Configuration specific properties Visibility F Hide Model Cglor. Suppression state C Suppression C Suppression C Suppression C Bigid C Brouble C	
Filde Model Cglor Suppression state C C Suppression C Solve as C C Bigid C	
Suppression state C Suppression C September C Breaker C Breaker C Breaker C Breaker C Breaker	
C Suppressed C Besolved Solve as C Bright C Brouble	
Solve as & Bigid C Elevable	
e Biğiq e Elevisie	
Reterenced configuration	
C Use component's "in-use" or last saved configuration	
C Use named configuration Standard	
1,5 MetBall	
Specify configurations to change Standard	
OK Cascel Help	

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

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



Clone the Project

8 Activate **Project 1** by using the Configuration Manager Tree.



Clone the Project

- 1 Click **FloWorks**, **Project**, **Clone Project**.
- 2 Click Add to existing.
- 3 In the **Existing configuration** list select **Project 2**.
- Click **OK**. Select **Yes** for each message dialog box that appears after you click **OK**.

Clone Project	? ×
C <u>C</u> reate new	
Add to existing	
Configuration <u>n</u> ame:	
Project 1 (1)	
Existing configuration:	
Project 2	
Copy results	
OK C	Cancel <u>H</u> elp

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

Analyze a Design Variant in the COSMOSFIoWorks Application

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

Activate the **Project 1** configuration.

Analyze a Design Variant in the COSMOSFIoWorks Application COSMOSFIoWorks 2004 Tutorial

- 1 Create a copy of the **Project 1** project by clicking FloWorks, Project, Clone Project.
- 2 Type Project 3 for the new project name and click **OK**.

Clone Project	? ×
C Add to existing	
Configuration name:	
Project 3	
Existing configuration:	
default	Ψ.
Copy results	
OK Cancel Help	

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

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