Model created in COMSOL Multiphysics 6.4

Ball Check Valve

Introduction

A ball check valve is a special type of valve in which a unidirectional fluid flow is ensured. In this valve, a movable spring-loaded ball controls the flow. When no inlet pressure is applied, the ball is forced into contact with an O-ring through the preload in the spring. For an inlet pressure imposing a flow in the operating direction, the slit between the ball and the O-ring opens when the fluid force acting on the ball becomes larger than the spring force. In the case of a flow in the opposite direction, the slit between the ball and the O-ring remains closed and the fluid is thus prevented from passing through the valve.

Figure 1 illustrates the fluid flow around the ball at the maximum opening position.

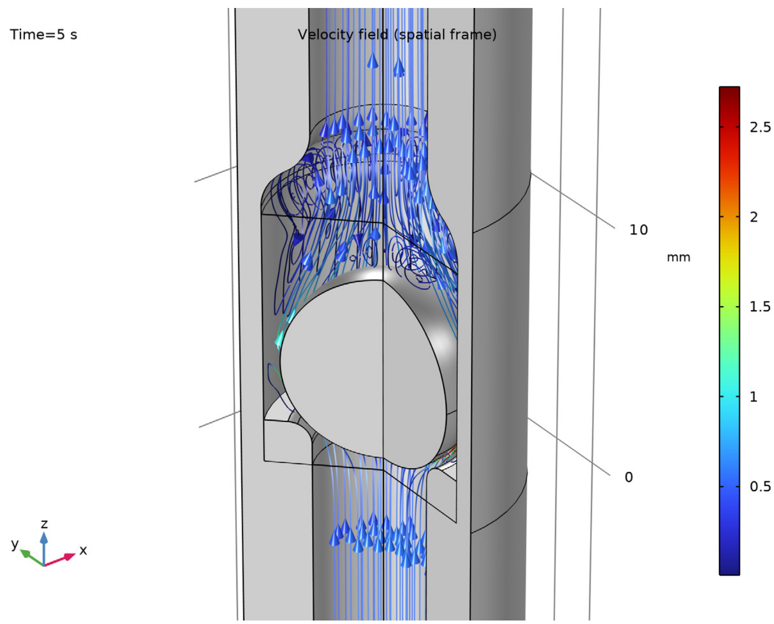


Figure 1: 3D view of the velocity field within the valve at the maximum opening position. The flow is directed upward in the figure.

In this example, you will learn how to solve a structural contact problem where a fluid is surrounding and acting on the potentially contacting solid parts.

Model Definition

The valve is 35 mm long with an outer diameter of 10 mm. The ball chamber inner diameter is 8.4 mm, and the ball diameter is 7.2 mm. The tube inner diameter is 5 mm. The ball chamber length is 12 mm.

The model geometry is reduced to a 2D axisymmetric cut as shown in Figure 2.

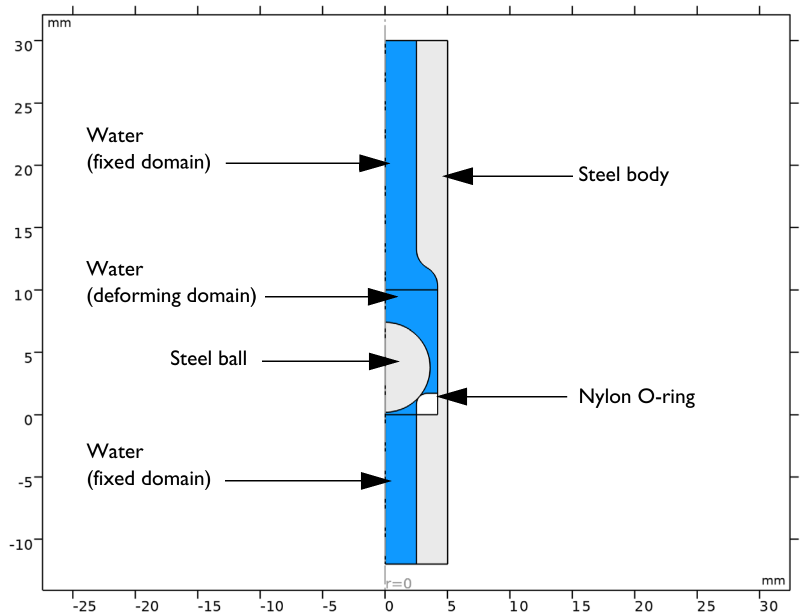


Figure 2: Geometry and materials.

MATERIAL PROPERTIES

The ball and the valve body are made of steel, the O-ring is made of nylon. The material properties are given in Table 1.

TABLE 1: VALVE MATERIAL PROPERTIES

Properties	**Structural steel**	**Nylon**
Density	7850 kg/m^3	1150 kg/m^3
Young's modulus	200 GPa	2 GPa
Poisson's ratio	0.3	0.4

The fluid used in this study is water at room temperature.

BOUNDARY CONDITIONS

Solid Mechanics

The ball is free to move along the symmetry axis and is subjected to both a spring load and to the fluid forces. The fluid forces are automatically applied when using the **Fluid-Structure Interaction** multiphysics coupling. The nylon O-ring is attached to the valve body which is assumed to be rigid. The rigidity is enforced by applying a **Fixed Constraint** to the entire valve body.

Structural contact between the ball and the O-ring is modeled.

The spring that holds the ball against the O-ring is not represented in the geometry, instead a spring foundation is used. The spring constant is 4 N/m, and the spring is under 5 mm predeformation when the ball is at rest in the valve. To ensure consistent initial conditions, the spring predeformation is ramped up using a smooth step function.

Turbulent Flow

To study the functioning of the valve, first a reversed flow and then an operating flow are applied. Upstream (bottom) and downstream (top) boundaries are defined as inlet and outlet conditions, respectively, with a varying pressure. The maximum pressure is 25 mbar.

Moving Mesh

The mesh surrounding the ball is set to deform freely, following a Yeoh mesh smoothing deformation. The mesh displacement is controlled by the structural displacement at the boundaries. At the remaining boundaries, adjacent to the fluid, a fixed boundary is used.

Results and Discussion

Figure 3 shows the fluid velocity at the maximum opening of the valve. The maximum velocity is about 2.7 m/s and the main flow path and the recirculation flow around it are clearly visible.

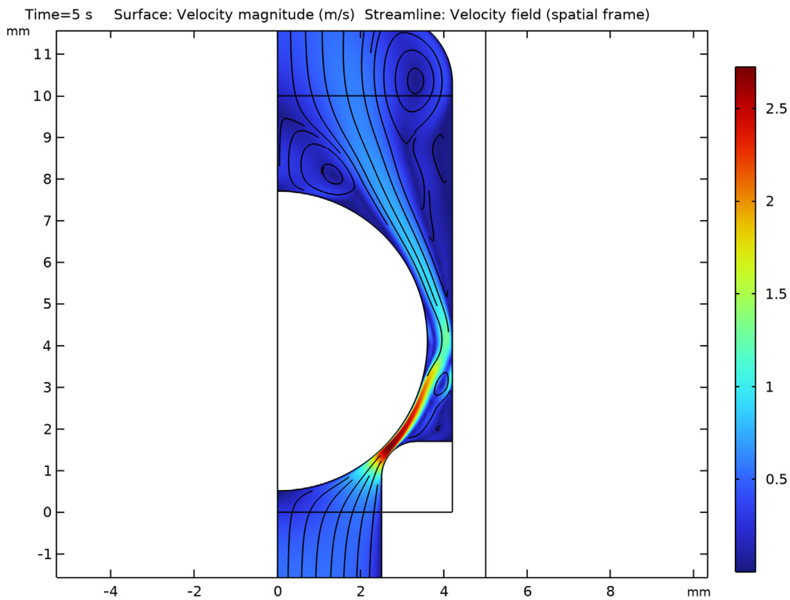


Figure 3: Fluid velocity at the maximum opening of the valve.

Figure 4 shows the pressure distribution in the fluid and the von Mises stress in the solid at maximum opening of the valve.

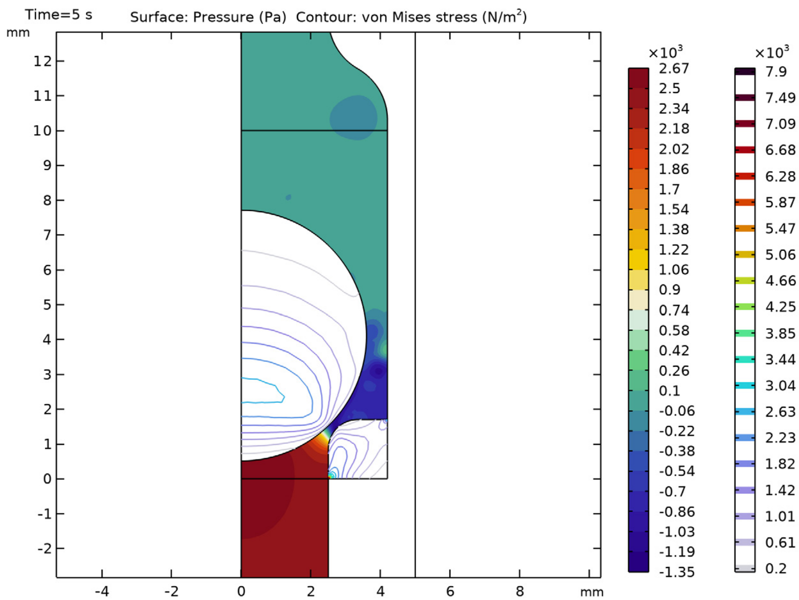


Figure 4: Fluid pressure and von Mises stress at maximum opening of the valve.

[Figure 5](#) and [Figure 6](#) show the fluid pressure when the valve is closed and under maximum pressure in the reverse direction.

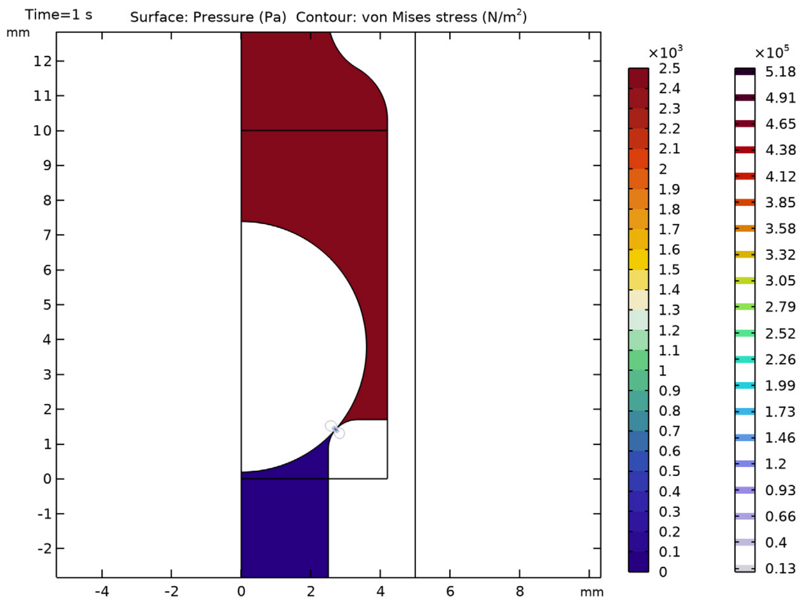


Figure 5: Fluid pressure and von Mises stress at maximum reversed pressure.

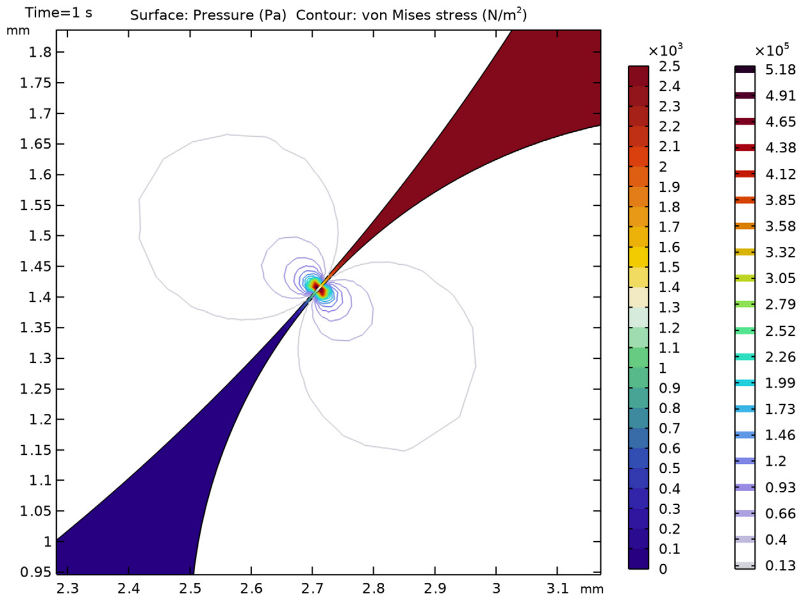


Figure 6: Fluid pressure and von Mises stress at maximum reverse pressure (close-up view of the contact region).

The fluid pressure drop when the flow is blocked in the contact region is clearly visible. The contact is almost localized as a single point contact.

Figure 7 shows the ball displacement as a function of time.

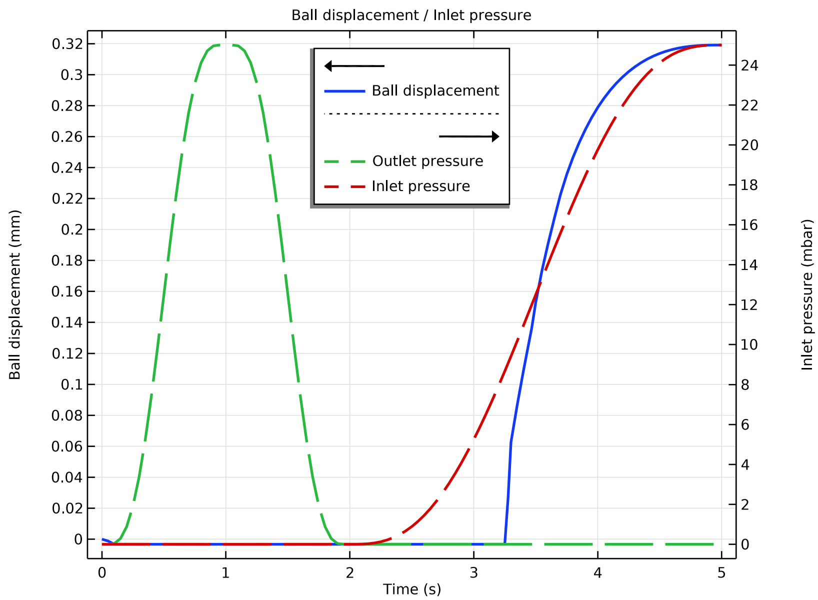


Figure 7: Ball center displacement versus time. The inlet pressures are shown for reference.

The ball moves down slightly as the spring predeformation is applied. As the reverse pressure increases, the displacement of the ball is negligible. After 2 s, the flow in the operating direction is increased until the ball reaches a maximum displacement of about 0.32 mm.

Figure 8 shows the flow in the valve as a function of time.

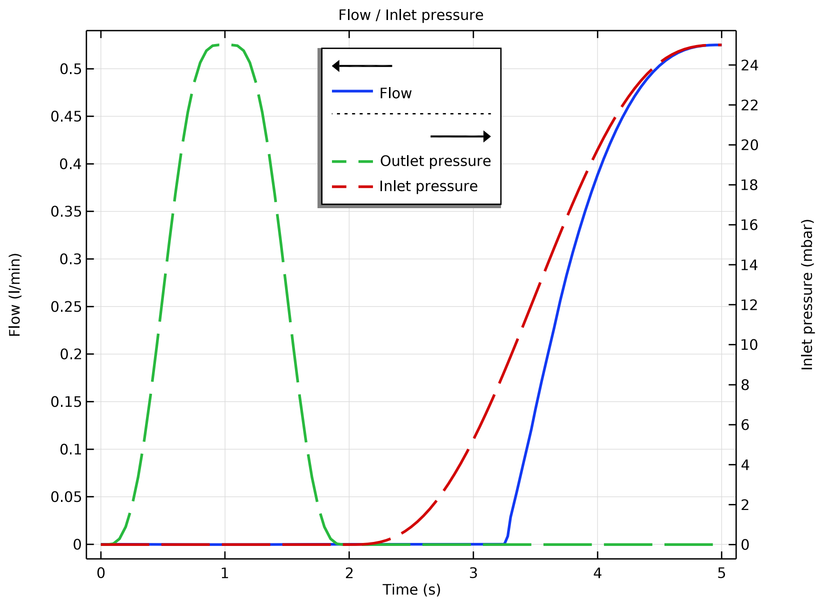


Figure 8: Fluid flow in the valve versus time.

Up to 1 s, the pressure is imposed in the reversed flow direction. The maximum reverse flow is about 0.3 ml/min. This small value is explained by the clearance between the parts in contact which is necessary to avoid topology changes in the fluid domain (see the section Notes About the COMSOL Implementation for more details). The reversed flow can be decreased even more by reducing the contact offset value, at the price of extra solution time caused by mesh refinement.

Figure 9 shows the operating curve of the valve.

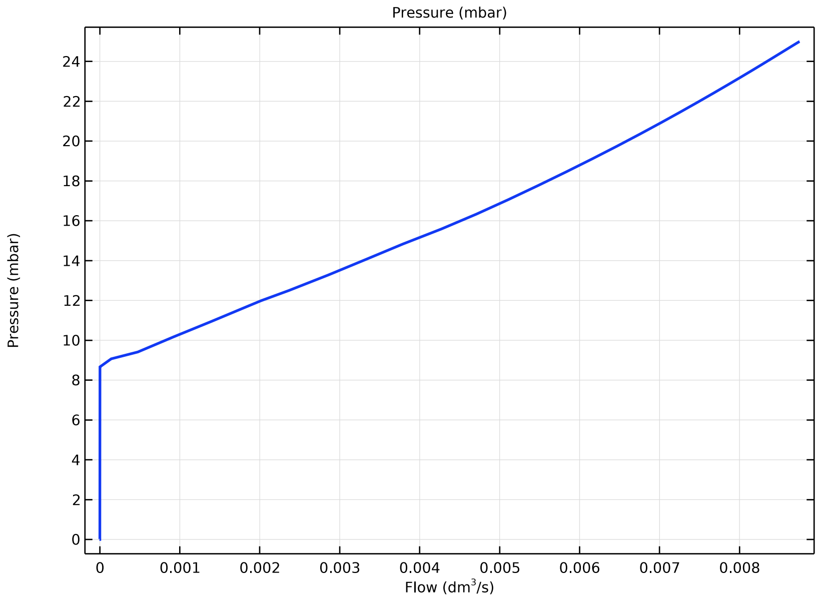


Figure 9: Operating curve of the valve.

From this plot, the opening pressure is seen to be about 9 mbar.

[Figure 10](#) shows the fluid pressure drop in the valve together with the inlet pressure conditions applied during the analysis.

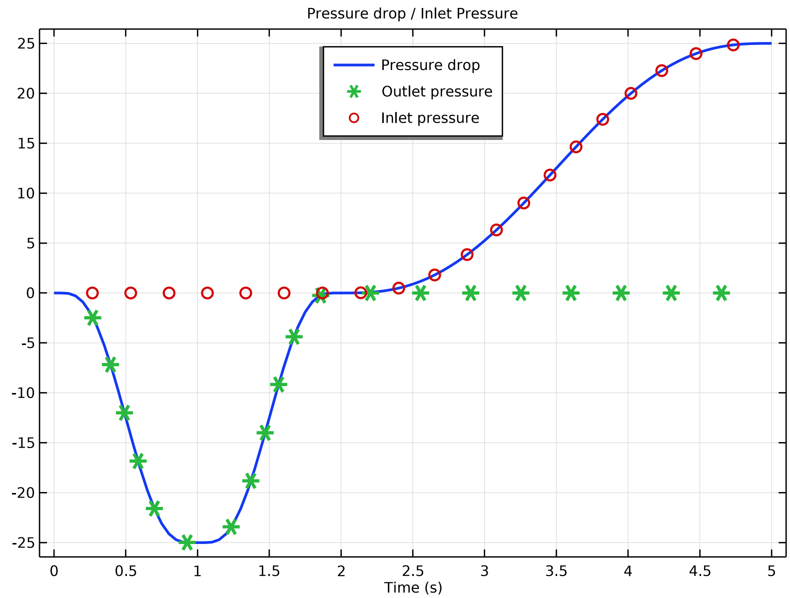
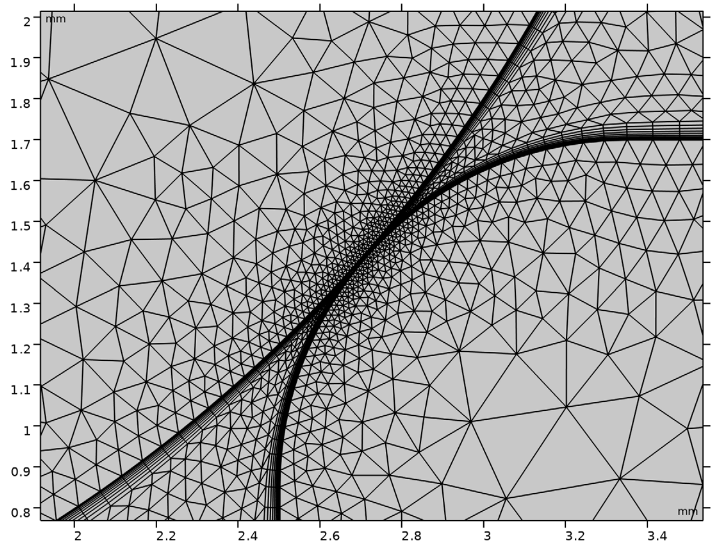


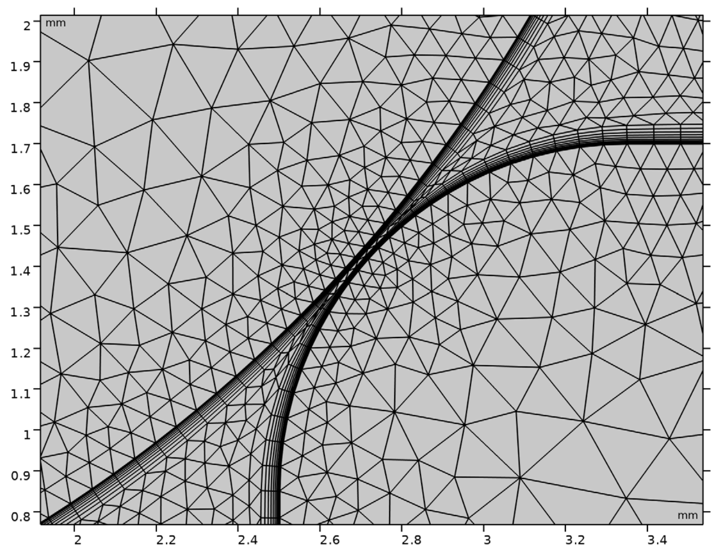
Figure 10: Pressure drop in the valve and inlet pressure condition at top boundary/ downstream (dashed green) and bottom/upstream boundary (dashed red).

Notes About the COMSOL Implementation

The main purpose of this model is to show how to solve a fluid structure interaction where two elastic parts separated by a fluid can get in contact with each other. The Arbitrary Lagrangian Eulerian (ALE) formulation used for the discretization of the Navier-Stokes equations in a deforming domain requires that the topology of this domain does not change. In a real contact situation, the topology of the fluid domain changes as the parts get into contact. For a numerical analysis, you need to include an offset in the contact settings to prevent the parts to get into physical contact. In this model, the offset is set to a very small value (5 µm) which is sufficient to preserve the fluid-domain topology while still preventing any significant flow in the reverse direction when the valve is closed.

To obtain a good accuracy of the solution, automatic remeshing is used when the ball is getting into contact with the O-ring or is moving away. Because of boundary layer mesh, it is recommended to use distortion as condition for remeshing, here the geometry is remeshed when the square root of the maximum element distortion exceeds 2.Figure 11 shows the different meshes generated by the Automatic Remeshing node during the computation.





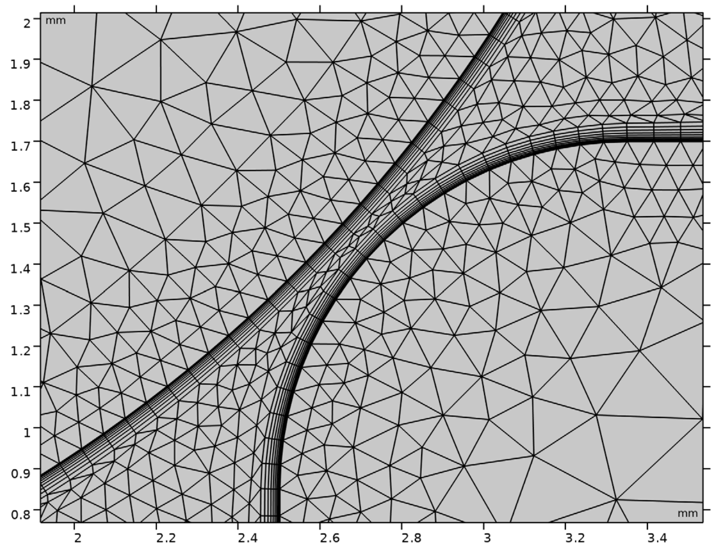


Figure 11: Mesh cases used during the computation (from top to bottom, left to right).

In order to ensure proper restart after the automatic remeshing, it is important to well refine the mesh at the fluid-structure interface. Moreover in the solver settings you can enforce consistent initialization.

The Fluid-Solid Interaction multiphysics interface adds a **Deforming Domain** node where you define the fluid domains to be controlled by structural deformation at their boundaries. In this model, the displacement of the ball is confined, and it is a good idea to split the fluid region into three domains. The central fluid domain (the one containing the moving ball) uses a freely moving deformed mesh, while the bottom and top fluid domains use a fixed mesh. This way you restrict the computation of the moving mesh equations to a minimum.

In this particular example, the valve body is actually modeled as rigid, so except for the contact region it could have been removed from the model entirely. Keeping it will, however, allow better visualization of the model and results.

In the Fluid-Structure Interaction multiphysics coupling, all boundaries between fluid and solid are selected as a default. You could choose to remove the boundaries adjacent to the rigid domains. Here the coupling type has instead been changed to **Fluid loading on structure** to avoid adding the extra degrees of freedoms that would be added for a bidirectional coupling in a fixed geometry case.

Application Library path: `Structural_Mechanics_Module/Fluid-Structure_Interaction/ball_check_valve`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **2D Axisymmetric**.

2 In the **Select Physics** tree, select **Fluid Flow** > **Single-Phase Flow** > **Turbulent Flow** > **Turbulent Flow, Algebraic yPlus (spf)**.

3 Click **Add**.

4 In the **Velocity field (m/s)** text field, type `u_fluid`.

5 In the **Velocity field components** table, enter the following settings:

u_fluid
v_fluid
w_fluid

6 In the **Select Physics** tree, select **Fluid Flow** > **Fluid–Structure Interaction** > **Fluid–Solid Interaction**.

7 Click **Add**.

8 In the **Added physics interfaces** tree, select **Laminar Flow (spf2)**.

9 Right-click and choose **Remove**.

10 Click **Study**.

11 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces** > **Turbulent Flow, Algebraic yPlus** > **Time Dependent with Initialization**.

12 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
`r1`	`2.5[mm]`	0.0025 m	`Inner radius`
`r2`	`4.2[mm]`	0.0042 m	`Ball chamber inner radius`
`r3`	`3.6[mm]`	0.0036 m	`Ball radius`
`r4`	`5[mm]`	0.005 m	`Outer radius`
`l`	`12[mm]`	0.012 m	`Ball chamber length`
`p0`	`25[mbar]`	2500 Pa	`Maximum inlet pressure`
`k0`	`4[N/m]`	4 N/m	`Spring constant`
`l0`	`5[mm]`	0.005 m	`Spring predeformation`
`offset`	`5[um]`	5E-6 m	`Contact offset`

GEOMETRY 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.

2 In the **Settings** window for **Geometry**, locate the **Units** section.

3 From the **Length unit** list, choose **mm**.

Rectangle 1 (r1)

1 In the **Geometry** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `r4-r1`.

4 In the **Height** text field, type `l`.

5 Locate the **Position** section. In the **r** text field, type `r1`.

6 In the **z** text field, type `-l`.

Rectangle 2 (r2)

1 In the **Geometry** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `r4-r2`.

4 In the **Height** text field, type `l`.

5 Locate the **Position** section. In the **r** text field, type `r2`.

Rectangle 3 (r3)

1 In the **Geometry** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `r4-r1`.

4 In the **Height** text field, type `3/2*1`.

5 Locate the **Position** section. In the **r** text field, type `r1`.

6 In the **z** text field, type `1`.

Union 1 (uni1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

2 Click in the **Graphics** window and then press Ctrl+A to select all objects.

3 In the **Settings** window for **Union**, locate the **Union** section.

4 Clear the **Keep interior boundaries** checkbox.

Fillet 1 (fil1)

1 In the **Geometry** toolbar, click **Fillet**.

2 On the object **uni1**, select Point 6 only.

3 In the **Settings** window for **Fillet**, locate the **Radius** section.

4 In the **Radius** text field, type `r2-r1`.

Fillet 2 (fil2)

1 In the **Geometry** toolbar, click **Fillet**.

2 On the object **fil1**, select Point 3 only.

3 In the **Settings** window for **Fillet**, locate the **Radius** section.

4 In the **Radius** text field, type `r2-r1`.

Square 1 (sq1)

1 In the **Geometry** toolbar, click **Square**.

2 In the **Settings** window for **Square**, locate the **Size** section.

3 In the **Side length** text field, type `r2-r1`.

4 Locate the **Position** section. In the **r** text field, type `r1`.

Fillet 3 (fil3)

1 In the **Geometry** toolbar, click **Fillet**.

2 On the object **sq1**, select Point 4 only.

3 In the **Settings** window for **Fillet**, locate the **Radius** section.

4 In the **Radius** text field, type `(r2-r1)/2`.

Rectangle 4 (r4)

1 In the **Geometry** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `r4`.

4 In the **Height** text field, type `7/2*1`.

5 Locate the **Position** section. In the **z** text field, type `-1`.

Circle 1 (c1)

1 In the **Geometry** toolbar, click **Circle**.

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `r3`.

4 In the **Sector angle** text field, type `180`.

5 Locate the **Position** section. In the **z** text field, type `r3`.

6 Locate the **Rotation Angle** section. In the **Rotation** text field, type `-90`.

You will now create imprints at the expected contact location.

Partition Domains 1 (pard1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Partition Domains**.

2 On the object **c1**, select Domain 1 only.

3 On the object **fil3**, select Domain 1 only.

4 In the **Settings** window for **Partition Domains**, locate the **Partition Domains** section.

5 Click to select the **Activate Selection** toggle button for **Vertices defining line segments**.

6 From the **Partition with** list, choose **Edges**.

7 On the object **c1**, select Boundary 1 only.

8 On the object **fil3**, select Boundary 5 only.

Move 1 (mov1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Move**.

2 Select the object **pard1(1)** only.

3 In the **Settings** window for **Move**, locate the **Displacement** section.

4 In the **z** text field, type `0.19` to ensure a clearance slightly above the contact offset.

Union 2 (uni2)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

2 Select the objects **mov1** and **pard1(2)** only.

3 In the **Settings** window for **Union**, locate the **Union** section.

4 Clear the **Keep interior boundaries** checkbox.

Rectangle 5 (r5)

1 In the **Geometry** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `r2`.

4 In the **Height** text field, type `10`.

With this fourth rectangle object, you will be able to define where to apply a moving mesh domain.

Form Union (fin)

1 In the **Model Builder** window, click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, click **Build Selected**.

3 Click the **Zoom Extents** button in the **Graphics** toolbar.

ADD MATERIAL

1 In the **Materials** toolbar, click **Add Material** to open the **Add Material** window.

2 Go to the **Add Material** window.

3 In the tree, select **Built-in** > **Water, liquid**.

4 Right-click and choose **Add to Component 1 (comp1)**.

5 In the tree, select **Built-in** > **Nylon**.

6 Right-click and choose **Add to Component 1 (comp1)**.

7 In the tree, select **Built-in** > **Structural steel**.

8 Right-click and choose **Add to Component 1 (comp1)**.

9 In the **Materials** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Nylon (mat2)

1 Select Domain 6 only.

2 In the **Settings** window for **Material**, click to expand the **Appearance** section.

3 From the **Material type** list, choose **Plastic**.

Structural steel (mat3)

1 In the **Model Builder** window, click **Structural steel (mat3)**.

2 Select Domains 3 and 5 only.

3 In the **Settings** window for **Material**, locate the **Appearance** section.

4 From the **Material type** list, choose **Steel**.

5 In the **Model Builder** window, click **Materials**.

6 In the **Settings** window for **Materials**, in the **Graphics** window toolbar, click next to **Colors**, then choose **Show Material Color and Texture**.

MOVING MESH

Deforming Domain 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Moving Mesh** click **Deforming Domain 1**.

2 Select Domain 2 only.

Prescribed Normal Mesh Displacement 1

1 In the **Moving Mesh** toolbar, click **Prescribed Normal Mesh Displacement**.

2 Select Boundaries 3 and 7 only.

DEFINITIONS

Piecewise 1 (pw1)

1 In the **Definitions** toolbar, click **Piecewise**.

2 In the **Settings** window for **Piecewise**, type `p_outlet` in the **Function name** text field.

3 Locate the **Definition** section. From the **Smoothing** list, choose **Continuous second derivative**.

4 From the **Transition zone** list, choose **Absolute size**.

5 In the **Size of transition zone** text field, type `0.45`.

6 Find the **Intervals** subsection. In the table, enter the following settings:

Start	End	Function
-1	0.5	0

Start	End	Function
`0.5`	`1.5`	`p0`
`1.5`	`5`	`0`

7 Locate the **Units** section. In the **Arguments** text field, type `s`.

8 In the **Function** text field, type `Pa`.

Piecewise 2 (pw2)

1 In the **Definitions** toolbar, click **Piecewise**.

2 In the **Settings** window for **Piecewise**, type `p_inlet` in the **Function name** text field.

3 Locate the **Definition** section. From the **Smoothing** list, choose **Continuous second derivative**.

4 From the **Transition zone** list, choose **Absolute size**.

5 In the **Size of transition zone** text field, type `1.5`.

6 Find the **Intervals** subsection. In the table, enter the following settings:

Start	End	Function
`0`	`3.5`	`0`
`3.5`	`7`	`p0`

7 Locate the **Units** section. In the **Arguments** text field, type `s`.

8 In the **Function** text field, type `Pa`.

Step 1 (step1)

1 In the **Definitions** toolbar, click f(X) **More Functions** and choose **Step**.

2 In the **Settings** window for **Step**, type `predef` in the **Function name** text field.

3 Locate the **Parameters** section. In the **Location** text field, type `0.5[s]`.

4 In the **To** text field, type `-10`.

5 Click to expand the **Smoothing** section. In the **Size of transition zone** text field, type `1`.

Integration 1 (intop1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 10 only.

Integration 2 (intop2)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Point**.

4 Select Point 7 only.

5 Locate the **Advanced** section. Clear the **Compute integral in revolved geometry** checkbox.

Variables 1

1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.

2 In the **Settings** window for **Variables**, locate the **Variables** section.

3 In the table, enter the following settings:

Name	Expression	Unit	Description
`flow`	`intop1(w_fluid)`	m^3/s	`Flow`

TURBULENT FLOW, ALGEBRAIC YPLUS (SPF)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Turbulent Flow, Algebraic yPlus (spf)**.

2 Select Domains 1, 2, and 4 only.

3 In the **Settings** window for **Turbulent Flow, Algebraic yPlus**, locate the **Turbulence** section.

4 From the **Wall treatment** list, choose **Low Re**.

Inlet 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Inlet**.

2 Select Boundary 2 only.

3 In the **Settings** window for **Inlet**, locate the **Boundary Condition** section.

4 From the list, choose **Pressure**.

5 Locate the **Pressure Conditions** section. In the p_0 text field, type `p_inlet(t)`.

6 Clear the **Suppress backflow** checkbox.

Outlet 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Outlet**.

2 Select Boundary 10 only.

3 In the **Settings** window for **Outlet**, locate the **Pressure Conditions** section.

4 In the p_0 text field, type `p_outlet(t)`.

5 Clear the **Suppress backflow** checkbox.

SOLID MECHANICS (SOLID)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

2 Select Domains 3, 5, and 6 only.

Fixed Constraint 1

1 In the **Physics** toolbar, click **Domains** and choose **Fixed Constraint**.

2 Select Domain 5 only.

Spring Foundation 1

1 In the **Physics** toolbar, click **Domains** and choose **Spring Foundation**.

2 Select Domain 3 only.

3 In the **Settings** window for **Spring Foundation**, locate the **Spring** section.

4 From the **Spring type** list, choose **Total spring constant**.

5 From the list, choose **Diagonal**.

6 Specify the $\mathbf{k}_{\mathrm{tot}}$ matrix as

0	0
0	k0

Predeformation 1

1 In the **Physics** toolbar, click **Attributes** and choose **Predeformation**.

2 In the **Settings** window for **Predeformation**, locate the **Spring Predeformation** section.

3 Specify the $\mathbf{u}_0$ vector as

0	r
predef(t)	z

You will now set up the contact condition between the ball and the O-ring.

DEFINITIONS

Contact Pair 1 (p1)

1 In the **Definitions** toolbar, click **Pairs** and choose **Contact Pair**.

2 Select Boundary 29 only.

3 In the **Settings** window for **Pair**, locate the **Destination Boundaries** section.

4 Click to select the **Activate Selection** toggle button.

5 Select Boundary 28 only.

SOLID MECHANICS (SOLID)

Contact 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Solid Mechanics (solid)** click **Contact 1**.

2 In the **Settings** window for **Contact**, click to expand the **Contact Surface Offset and Adjustment** section.

3 In the $d_{\mathrm{offset,d}}$ text field, type `offset` to prevent the part from getting physically in contact. This way, the topology of the moving mesh domain is kept.

MULTIPHYSICS

Fluid–Structure Interaction 1 (fsi1)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Multiphysics** click **Fluid–Structure Interaction 1 (fsi1)**.

2 In the **Settings** window for **Fluid–Structure Interaction**, locate the **Coupling Type** section.

3 From the **Fixed geometry coupling type** list, choose **Fluid loading on structure**.

MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

3 From the **Element size** list, choose **Fine**.

4 Click **Build All**.

STUDY 1

Step 2: Time Dependent

1 In the **Model Builder** window, under **Study 1** click **Step 2: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

3 In the **Output times** text field, type `range(0,5e-2,5)`.

4 Click to expand the **Results While Solving** section. Select the **Plot** checkbox.

5 From the **Update at** list, choose **Time steps taken by solver**.

6 Click to expand the **Study Extensions** section. Select the **Automatic remeshing** checkbox.

Solution 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

3 In the **Model Builder** window, expand the **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Dependent Variables 2** node, then click **Pressure (comp1.p)**.

4 In the **Settings** window for **Field**, locate the **Scaling** section.

5 From the **Method** list, choose **Manual**.

6 In the **Scale** text field, type `1e3`.

7 In the **Model Builder** window, under **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Dependent Variables 2** click **Spatial Mesh Displacement (comp1.spatial.disp)**.

8 In the **Settings** window for **Field**, locate the **Scaling** section.

9 In the **Scale** text field, type `1e-3`.

10 In the **Model Builder** window, under **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Dependent Variables 2** click **Velocity Field (Spatial Frame) (comp1.u_fluid)**.

11 In the **Settings** window for **Field**, locate the **Scaling** section.

12 From the **Method** list, choose **Manual**.

13 In the **Scale** text field, type `1`.

14 In the **Model Builder** window, under **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Dependent Variables 2** click **Displacement Field (comp1.u_solid)**.

15 In the **Settings** window for **Field**, locate the **Scaling** section.

16 In the **Scale** text field, type `1e-3`.

17 In the **Model Builder** window, under **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** click **Time-Dependent Solver 1**.

18 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.

19 From the **Steps taken by solver** list, choose **Intermediate**.

20 In the **Model Builder** window, expand the **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Time-Dependent Solver 1** node, then click **Automatic Remeshing**.

21 In the **Settings** window for **Automatic Remeshing**, locate the **Condition for Remeshing** section.

22 From the **Condition type** list, choose **Distortion**.

23 Locate the **Remesh** section. From the **Consistent initialization** list, choose **Backward Euler**.

24 Click **Run**.

RESULTS

Velocity (spf)

The default plot shows the fluid velocity inside the valve at maximum opening.

Streamline 1

1 In the **Model Builder** window, right-click **Velocity (spf)** and choose **Streamline**.
2 In the **Settings** window for **Streamline**, locate the **Streamline Positioning** section.
3 From the **Positioning** list, choose **Uniform density**.
4 In the **Density level** text field, type `9.4`.
5 In the **Velocity (spf)** toolbar, click **Plot**.

Contour 1

1 In the **Model Builder** window, right-click **Pressure (spf)** and choose **Contour**.
2 In the **Settings** window for **Contour**, locate the **Expression** section.
3 In the **Expression** text field, type `solid.mises`.
4 Locate the **Coloring and Style** section. From the **Color table** list, choose **PrismDark**.
5 In the **Pressure (spf)** toolbar, click **Plot**.

Flow

1 In the **Results** toolbar, click **1D Plot Group**.
2 In the **Settings** window for **1D Plot Group**, type `Flow` in the **Label** text field.
3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Remeshed Solution 1 (sol3)**.
4 Locate the **Legend** section. Clear the **Show legends** checkbox.

Global 1

1 Right-click **Flow** and choose **Global**.
2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.
3 In the table, enter the following settings:

Expression	Unit	Description
`flow`	`l/min`	`Flow`

4 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.
5 In the **Title** text area, type `Flow (solid)`.
6 Click to expand the **Coloring and Style** section. From the **Width** list, choose **2**.

Global 2

1 In the **Model Builder** window, right-click **Flow** and choose **Global**.
2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.
3 In the table, enter the following settings:

Expression	Unit	Description
`p_outlet(t)`	`mbar`	`Outlet pressure`

4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.
5 From the **Width** list, choose **2**.

Global 3

1 Right-click **Flow** and choose **Global**.
2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.
3 In the table, enter the following settings:

Expression	Unit	Description
`p_inlet(t)`	`mbar`	`Inlet pressure`

4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.
5 From the **Width** list, choose **2**.
6 Locate the **Title** section. From the **Title type** list, choose **None**.

Flow

1 In the **Model Builder** window, click **Flow**.
2 In the **Settings** window for **1D Plot Group**, click to expand the **Title** section.
3 From the **Title type** list, choose **Manual**.
4 In the **Title** text area, type `Flow / Inlet pressure`.
5 Locate the **Plot Settings** section. Select the **Two y-axes** checkbox.
6 In the table, select the **Plot on secondary y-axis** checkboxes for **Global 2** and **Global 3**.
7 Select the **Secondary y-axis label** checkbox. In the associated text field, type `Inlet pressure (mbar)`.
8 Locate the **Legend** section. Select the **Show legends** checkbox.
9 From the **Position** list, choose **Upper middle**.
10 In the **Flow** toolbar, click **Plot**.

Ball Displacement

1 In the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Ball Displacement` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Remeshed Solution 1 (sol3)**.

Point Graph 1

1 Right-click **Ball Displacement** and choose **Point Graph**.

2 Select Point 4 only.

3 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `w_solid`.

5 Select the **Description** checkbox. In the associated text field, type `Ball displacement`.

6 Click to expand the **Coloring and Style** section. From the **Width** list, choose **2**.

7 Click to expand the **Legends** section. Select the **Show legends** checkbox.

8 From the **Legends** list, choose **Manual**.

9 In the table, enter the following settings:

Legends
Ball displacement

Global 2, Global 3

1 In the **Model Builder** window, under **Results** > **Flow**, Ctrl-click to select **Global 2** and **Global 3**.

2 Right-click and choose **Copy**.

Ball Displacement

1 In the **Model Builder** window, under **Results** right-click **Ball Displacement** and choose **Paste Multiple Items**.

2 In the **Settings** window for **1D Plot Group**, locate the **Title** section.

3 From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Ball displacement / Inlet pressure`.

5 Locate the **Plot Settings** section. Select the **Two y-axes** checkbox.

6 Select the **Secondary y-axis label** checkbox. In the associated text field, type `Inlet pressure (mbar)`.

7 In the table, select the **Plot on secondary y-axis** checkboxes for **Global 2** and **Global 3**.

8 Locate the **Legend** section. From the **Position** list, choose **Upper middle**.

9 In the **Ball Displacement** toolbar, click **Plot**.

Operating Curve

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Operating Curve` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Remeshed Solution 1 (sol3)**.

4 Locate the **Legend** section. Clear the **Show legends** checkbox.

Global 1

1 Right-click **Operating Curve** and choose **Global**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`p_inlet(t)`	`mbar`	`Pressure`

4 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

5 In the **Expression** text field, type `flow`.

6 From the **Unit** list, choose **dm^3/s**.

7 Locate the **Coloring and Style** section. From the **Width** list, choose **2**.

8 In the **Operating Curve** toolbar, click **Plot**.

Pressure Drop

1 In the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Pressure Drop` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Remeshed Solution 1 (sol3)**.

4 Locate the **Title** section. From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type `Pressure drop / Inlet Pressure`.

6 Locate the **Legend** section. From the **Position** list, choose **Upper middle**.

Point Graph 1

1 Right-click **Pressure Drop** and choose **Point Graph**.

2 Select Point 1 only.

3 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `p-intop2(p)`.

5 From the **Unit** list, choose **mbar**.

6 Select the **Description** checkbox. In the associated text field, type `Pressure drop`.

7 Locate the **Coloring and Style** section. From the **Width** list, choose **2**.

8 Locate the **Legends** section. Select the **Show legends** checkbox.

9 From the **Legends** list, choose **Manual**.

10 In the table, enter the following settings:

Legends
Pressure drop

Global 2, Global 3

1 In the **Model Builder** window, under **Results** > **Flow**, Ctrl-click to select **Global 2** and **Global 3**.

2 Right-click and choose **Copy**.

Pressure Drop

In the **Model Builder** window, under **Results** right-click **Pressure Drop** and choose **Paste Multiple Items**.

Global 2

1 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

2 In the table, enter the following settings:

Expression	Unit	Description
-p_outlet(t)	mbar	Outlet pressure

3 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

4 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

5 From the **Positioning** list, choose **Interpolated**.

6 In the **Number** text field, type `20`.

Global 3

1 In the **Model Builder** window, click **Global 3**.

2 In the **Settings** window for **Global**, locate the **Coloring and Style** section.

3 Find the **Line style** subsection. From the **Line** list, choose **None**.

4 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

5 From the **Positioning** list, choose **Interpolated**.

6 In the **Number** text field, type `20`.

7 In the **Pressure Drop** toolbar, click **Plot**.

In the following steps you will generate the plot shown in Figure 1.

Surface

1 In the **Model Builder** window, expand the **Velocity, 3D (spf)** node, then click **Surface**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `solid.disp`.

4 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

5 From the **Color** list, choose **Gray**.

6 Click to expand the **Title** section. From the **Title type** list, choose **None**.

Streamline 1

1 In the **Model Builder** window, right-click **Velocity, 3D (spf)** and choose **Streamline**.

2 In the **Settings** window for **Streamline**, locate the **Streamline Positioning** section.

3 From the **Positioning** list, choose **Uniform density**.

4 In the **Density level** text field, type `11.5`.

5 Locate the **Coloring and Style** section. Find the **Point style** subsection. From the **Type** list, choose **Arrow**.

Color Expression 1

Right-click **Streamline 1** and choose **Color Expression**.

Velocity, 3D (spf)

In the **Velocity, 3D (spf)** toolbar, click **Plot**.