Created in COMSOL Multiphysics 6.0

Water Hammer

Introduction

When a valve is closed rapidly in a pipe network it gives rise to a hydraulic transient - a pressure wave - known as a water hammer. The propagation of these hydraulic transients can in extreme cases cause failures of pipe systems due to the created overpressures (see Ref. 1). This is a model of a simple verification pipe system consisting of a reservoir, a pipe, and a valve (see Ref. 2). In this model, the valve is closed instantaneously.

Model Definition

The model consists of a reservoir connected to a pipe of length $L = 20$ m with an inner radius $R = 398.5$ mm. A valve is located at the other end of the pipe. The model is sketched in the figure below.

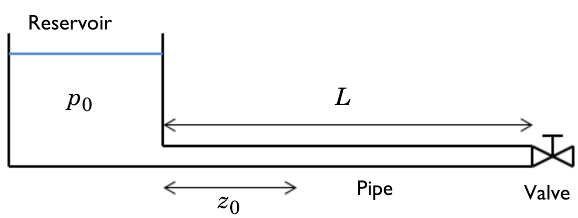


Figure 1: Pipe system with reservoir and valve.

The pipe is made of steel with Young's modulus E = 210 GPa and wall thickness w = 8 mm. At a distance z_0 = 11.15 m from the reservoir there is a pressure sensor measurement point. The reservoir acts as a constant pressure source with p_0 = 1 atm. As an initial condition, the valve is open and water is flowing steadily at a flow rate of Q_0 = 0.5 m^3/s. At time t = 0 s the valve is closed instantaneously, thereby initiating the water hammer.

As a result of the compressibility of the water and the elastic behavior of the pipe, a sharp pressure pulse is generated traveling upstream of the valve. The water hammer wave speed c is given by the expression

$$\frac{1}{c^2} = \frac{1}{c_s^2} + \rho\beta_A$$

where c_s is the isentropic speed of sound in the bulk fluid (1481 m/s for water), while the second term is caused by the elasticity of the pipe walls. The water density is ρ, and β_A is the pipe cross-sectional compressibility, and the resulting effective wave speed is 1037 m/s.

The instantaneous closure of the valve results in a water hammer pulse of amplitude P given by Joukowsky's fundamental equation (see Ref. 1)

$$P = \rho c u_o \tag{1}$$

where u_0 is the average fluid velocity before valve closure.

Notes About the COMSOL Implementation

Because the valve is assumed to close instantaneously, the generated water hammer pulse has a step function like shape. This must be resolved numerically and it thus requires a well-posed numerical scheme to correctly solve the problem.

The length of the pipe is meshed with N elements giving a mesh size $dx = L/N$. For the transient solver to be well behaved, changes of the pressure during a time step dt cannot move fully across a mesh element. That is, the distance traversed by the pressure signal during dt, that distance being $c{\cdot}dt$, must be smaller than the typical mesh size dx. This is captured by imposing the following CFL number condition

$$\mathrm{CFL} = 0.1 = \frac{c \cdot dt}{dx} \tag{2}$$

meaning that changes during the time dt maximally move 10% of the mesh length dx. The upshot is that the resolution of space (via the mesh resolution) and time (via the solver time step) are interdependent, and thus cannot be changed independently: increasing the mesh resolution requires decreasing the time stepping.

Even when satisfying the CFL condition of Equation 2, the numerical solution close to the region of the instantaneously changing pressure will exhibit spurious and nonphysical oscillations known as Gibbs oscillations. These are due to the inherent mathematical difficulty of resolving a discontinuously changing function by a finite number of mesh elements; it is similar to the oscillations observed around discontinuities when using a truncated Fourier series to approximate the underlying function. The oscillations can be restricted by adding some amount of high frequency damping in the generalized-α time dependent solver. This is done in the model and described below in the instructions.

Results and Discussion

The excess pressure history, $p - p_0$, as measured at the pressure sensor located at z_0 is shown in Figure 2. The curves correspond very well to the results obtained in the verification model of Ref. 2 (visual comparison) and thus verify the water hammer model. Notice the numerically-induced high-frequency oscillations around the discontinuities in the pressure (the abrupt changes). As described in Notes About the COMSOL Implementation these are numerical artifacts with no physical meaning which should be ignored when evaluating the results in this and the following figures. The ripples can be reduced by increasing the mesh resolution parameter N. This in turn decreases the time step dt prescribed by Equation 2 and results in longer computational times.

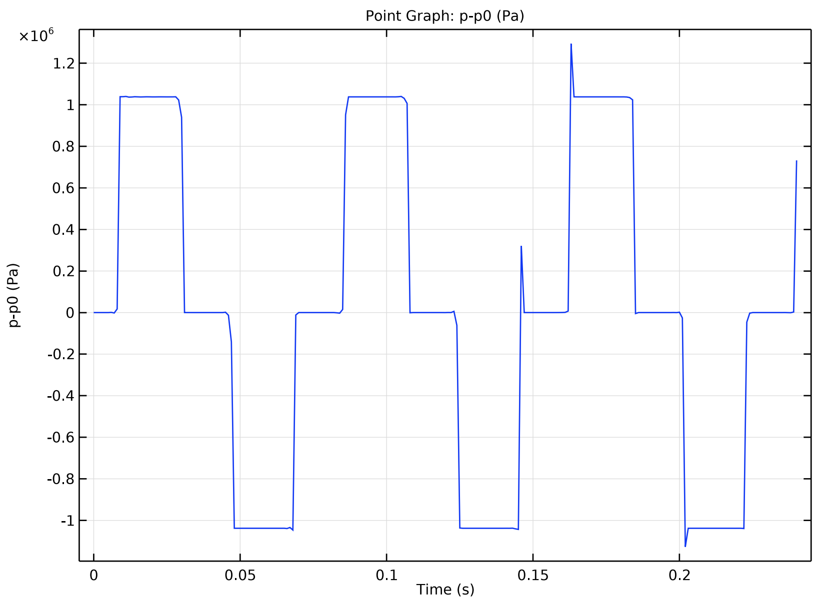


Figure 2: Excess pressure history measured at the pressure sensor.

The excess pressure at the valve is plotted by the blue line in Figure 3 together with the water hammer amplitude predicted by Equation 1 (green line). The amplitude of the excess water hammer pressure match perfectly with the theoretical prediction for the positive oscillations. This is not surprising as the theory of Joukowsky is based in lossless sudden closure of a valve.

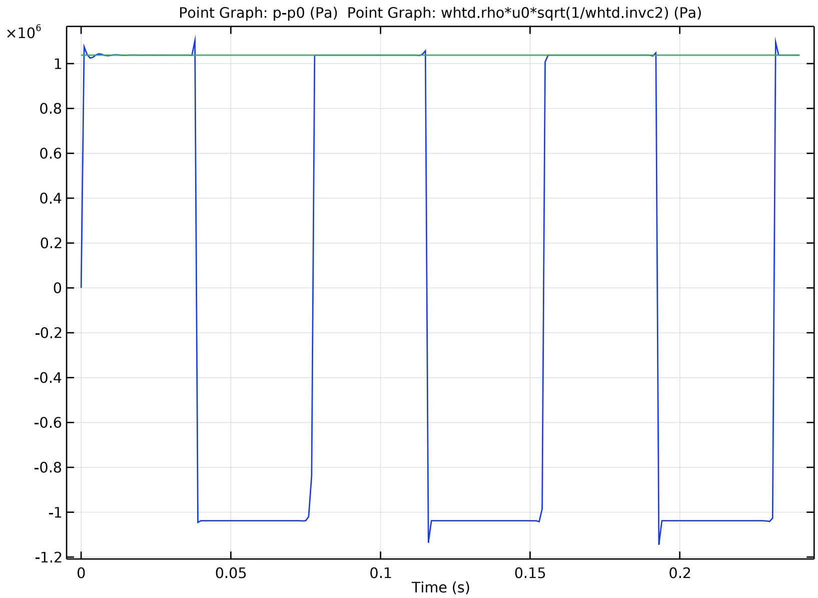


Figure 3: Excess pressure at the valve (blue line) and predicted water hammer amplitude (green line).

The final plot shown in Figure 4 illustrates the pressure distribution along the pipe at time t = 0.24 s.

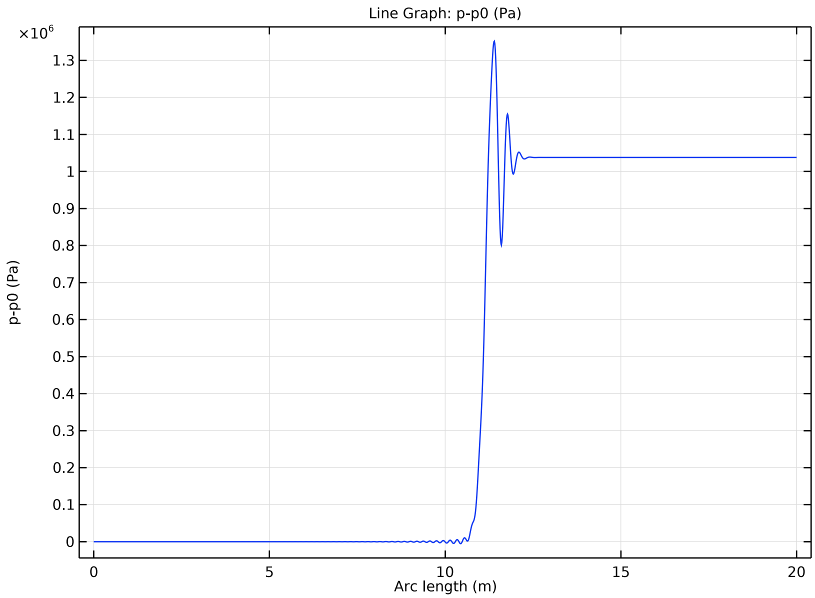


Figure 4: Excess pressure distribution along the pipe for t = 0.24 s.

References

1. M.S. Ghidaoui, M. Zhao, D.A. McInnis, and D.H. Axworthy, “A Review of Water Hammer Theory and Practice,” *Applied Mechanics Reviews*, ASME, 2005.

2. A.S. Tijsseling, “Exact Solution of Linear Hyperbolic Four-Equation Systems in Axial Liquid-Pipe Vibration,” *J. Fluids and Structures*, vol. 18, pp 179–196, 2003.

Application Library path: `Pipe_Flow_Module/Verification_Examples/water_hammer_verification`

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 **3D**.

2 In the **Select Physics** tree, select **Fluid Flow>Single-Phase Flow>Water Hammer (whtd)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies>Time Dependent**.

6 Click **Done**.

GLOBAL DEFINITIONS

Load the parameters for the model. The list of parameters includes the pipe properties, the initial flow values, and the mesh and time step parameters.

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 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `water_hammer_verification_parameters.txt`.

GEOMETRY 1

Polygon 1 (pol1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Polygon**.

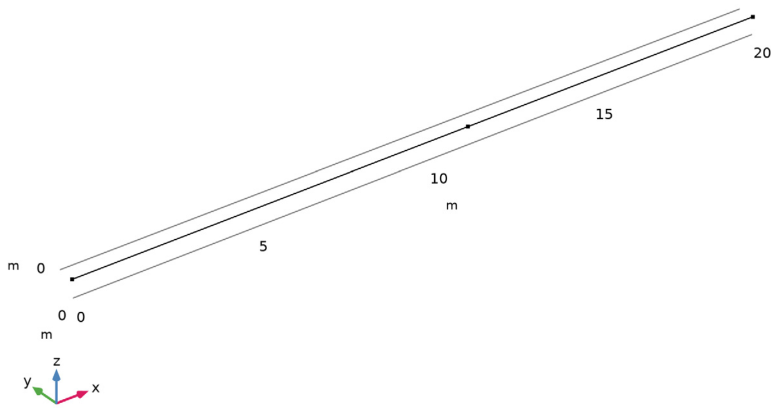
2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 In the table, enter the following settings:

x (m)	y (m)	z (m)
0	0	0
z0	0	0
L	0	0

4 Click **Build All Objects**.

The geometry should look like the figure below.



ADD MATERIAL

1. In the **Home** 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. Click **Add to Component** in the window toolbar.
5. In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

WATER HAMMER (WHTD)

Fluid Properties 1

All the fluid properties are selected automatically and the material parameters are retrieved from the water material just added. Proceed to set up the properties of the pipe (shape and material) and disable any flow resistance models.

Pipe Properties 1

1. In the **Model Builder** window, click **Pipe Properties 1**.
2. In the **Settings** window for **Pipe Properties**, locate the **Pipe Shape** section.

3 From the list, choose **Circular**.

4 In the d_i text field, type `2*R`.

5 Locate the **Pipe Model** section. From the E list, choose **User defined**. In the associated text field, type `E`.

6 In the Δw text field, type `w`.

7 Locate the **Flow Resistance** section. From the **Friction model** list, choose **User defined**.

Initial Values 1

1 In the **Model Builder** window, click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 In the p text field, type `p0`.

4 In the u text field, type `u0`.

Pressure 1

1 In the **Physics** toolbar, click **Points** and choose **Pressure**.

2 Select Point 1 only.

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

4 In the p_{in} text field, type `p0`.

Next build the mesh and set the mesh size to `L/N`, where `N` is a value defined in the parameters.

MESH 1

Edge 1

1 In the **Mesh** toolbar, click **Boundary** and choose **Edge**.

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

Size

1 In the **Model Builder** window, click **Size**.

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

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `L/N`.

5 In the **Minimum element size** text field, type `1[mm]`.

6 Click **Build All**.

STUDY 1

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 1** click **Step 1: Time Dependent**.
2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.
3 In the **Output times** text field, type `range(0,1e-3,0.24)`.

Before solving the model, generate the default solver sequence in order to set the maximal time step to the desired value `dt`, Equation 2. Note that the solving process may take a couple of minutes.

Solution 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.
2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.
3 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.
4 From the **Steps taken by solver** list, choose **Manual**.
5 In the **Time step** text field, type `dt`.
6 In the **Amplification for high frequency** text field, type `0.2`.

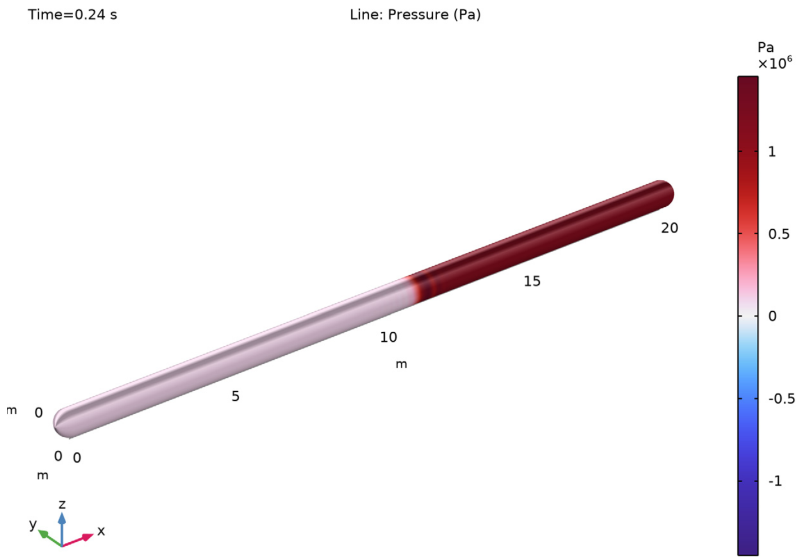
   Changing the alpha parameter to a lower value will result in more damping of the unwanted high-frequency Gibbs oscillations in the solution.

   An alternative solver strategy can be to switch to the BDF solver of order one and use a very small manual time step, for example, setting the CFL number to 0.01. Notice that this solver is very diffusive and will smooth the shock but it will avoid the oscillations.
7 In the **Study** toolbar, click **Compute**.

RESULTS

Acoustic Pressure (whtd)

The pressure along the pipe at time $t = 0.24$ s looks as in the figure below.



1D Plot Group 3

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

Point Graph 1

1. Right-click **1D Plot Group 3** and choose **Point Graph**.
2. Select Point 2 only.
3. In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.
4. In the **Expression** text field, type `p-p0`.
5. In the **1D Plot Group 3** toolbar, click **Plot**.

Measurement point

1. In the **Model Builder** window, right-click **1D Plot Group 3** and choose **Rename**.
2. In the **Rename 1D Plot Group** dialog box, type `Measurement point` in the **New label** text field.
3. Click **OK**.

The excess pressure history, $p - p_0$, measured at the probe point z_0 is depicted in Figure 3.

1D Plot Group 4

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

Point Graph 1

1. Right-click **1D Plot Group 4** and choose **Point Graph**.
2. Select Point 3 only.
3. In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.
4. In the **Expression** text field, type `p-p0`.

Point Graph 2

1. Right-click **Point Graph 1** and choose **Duplicate**.
2. In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.
3. In the **Expression** text field, type `whtd.rho*u0*sqrt(1/whtd.invc2)`.
4. In the **1D Plot Group 4** toolbar, click **Plot**.

Valve

1. In the **Model Builder** window, right-click **1D Plot Group 4** and choose **Rename**.
2. In the **Rename 1D Plot Group** dialog box, type `Valve` in the **New label** text field.
3. Click **OK**.

   Figure 3 shows the excess pressure $p - p_0$ at the valve together with the pressure rise predicted by Joukowsky's equation, see Equation 1.

1D Plot Group 5

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

Line Graph 1

1. Right-click **1D Plot Group 5** and choose **Line Graph**.
2. Click in the **Graphics** window and then press Ctrl+A to select both edges.
3. In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.
4. In the **Expression** text field, type `p-p0`.

Pressure profile

1. In the **Model Builder** window, click **1D Plot Group 5**.
2. In the **Settings** window for **1D Plot Group**, locate the **Data** section.
3. From the **Time selection** list, choose **Last**.
4. In the **1D Plot Group 5** toolbar, click **Plot**.

5 Right-click **1D Plot Group 5** and choose **Rename**.

6 In the **Rename 1D Plot Group** dialog box, type `Pressure profile` in the **New label** text field.

7 Click **OK**.

The excess pressure $p - p_0$ along the pipe at t = 0.24 s is depicted in Figure 4. The numerically induced ripples on the pressure profile can be reduced by increasing the mesh resolution parameter `N`. Increasing this parameter will reduce the time step `dt` according to Equation 2 and thus increase the total computational time.