KAPITEL 10 / *CHAPTER 10* [11]
PROSPECTS OF USING NUMERICAL SIMULATION SOFTWARE FOR WATER HAMMER ANALYSIS AND RESEARCH

DOI: 10.30890/2709-2313.2023-24-01-005

Introduction

Before there was a word for it, water hammer was being used in Roman public water supply pipes and stone tubes as early as the 1st century BC. Nasser al-Ahmar built the Alhambra in Granada beginning in AD1238 using a hydram. The hydram pumped water through a first reservoir, which was filled with water from a channel in the Darro River. The water then emptied into a large vertical channel that led to a second reservoir beneath. The whirlpool created by the whirlpool propelled water through a smaller pipe up to six metres, with most of the water draining into a second slightly larger pipe.

John Whitehurst, an Englishman, constructed a hydraulic ram in 1772 for a house in Cheshire. Joseph Michel Montgolfier (1740–1810), a French inventor, constructed a hydraulic ram for his Voiron paper mill in 1796. The hydraulic ram is the source of the terms "water hammer" in both French and Italian, which translate to "blow of the ram" (Coup de Bélier in French and Colpo d'Ariete in Italian). Civil engineers started to worry about water hammer as municipal water systems were installed in the 19th century. In Physiologists who studied the vascular system were likewise intrigued by the water hammer [1].

The notion of water hammer is commonly regarded as having started in 1883 with the work of German physiologist Johannes von Kries (1853-1928), who was studying the pulse in blood arteries, even though it was anticipated in work by Thomas Young. In Civil engineers, however, ignored his conclusions. In 1898, Russian fluid dynamicist Nikolay Yegorovich Zhukovsky (1847-1921) independently deduced Kries's conclusions.The American civil engineer Joseph Palmer Frizell (1832–1910) completed in 1898. The Italian engineer Lorenzo Allievi (1856–1941) completed [1].

[11] ***Authors:*** *Mysak Ihor Vasylovych*

 ISBN 978-3-98924-006-3

10.1. Modern importance of water hammer research.

Hydraulic shock, also known as a fluid hammer or water hammer, is a pressure wave or surge that results from a fluid in motion - typically a liquid, but occasionally also a gas-being abruptly forced to stop or reverse direction, causing a shift in momentum. This occurrence frequently happens when a pipeline system's end valve abruptly closes, causing a pressure wave to spread throughout the pipe [2].

Major issues could result from this pressure wave, including noise and vibration as well as pipe failure or rupture. With the use of accumulators, expansion tanks, surge tanks, blowoff valves, and other features, the consequences of the water hammer pulses can be lessened (fig 1). Making sure that no valves stop too quickly while there is a considerable flow will prevent the impacts, although there are other circumstances that can impact.

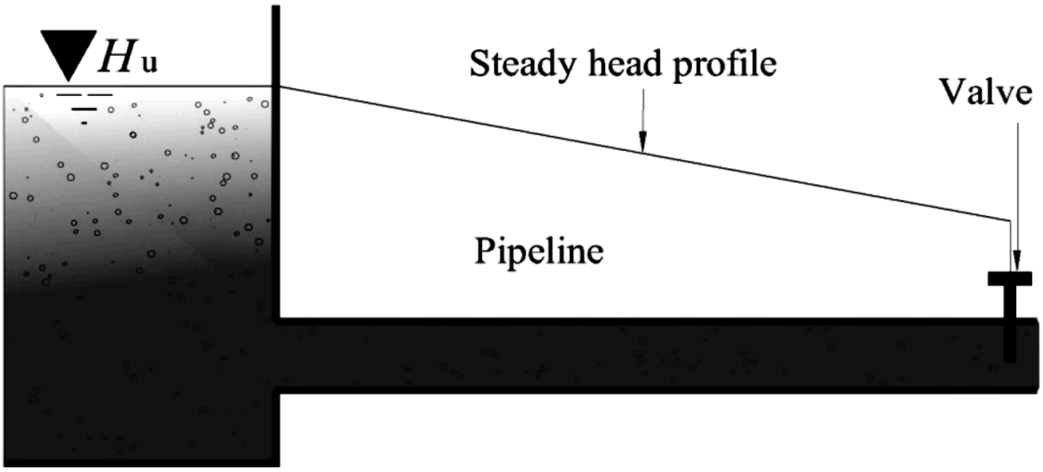

Fig. 1 - Schematic diagram of the occurrence of water hamer

The majority of engineers who plan pumping systems are familiar with the words "water hammer," "surge pressure," and hydraulic transient. It is less easy to determine whether or not a transient flow or surge study is required during the planning stage. In unfavorable conditions, pipelines longer than one hundred meters and carrying merely a few tenths of a liter per second may sustain damage from water hammer [3]. However, if they are not adequately secured, resonant vibrations in pumping stations can damage even extremely short, unsupported pipelines. In contrast, the occurrence is less frequent in building services systems, such as drinking water supply and heating pipelines, which are usually small in length and have a a small cross-section.

Closing the valve too quickly downstream of the pump can cause water hammer

- an effect that can lead to the destruction of the filter housing, pipeline, hydraulic accumulator, connections, and valves due to the inertial movement of water. We have all heard signs of water hammer in apartments of multi-story buildings - noise, knocking, and rattling in pipelines. If water moves in a pipeline at a speed of 3 meters per second, and a valve downstream is suddenly closed, a shock wave with a pressure of up to 70 bar occurs. This wave travels at the speed of sound (about 1500 m/s), is reflected from an obstacle, and moves back until it comes into contact with an obstacle - a pump, check valve, or filter housing [4]. Such an impulse can easily destroy the charging filter housing, for example, by creating a vacuum in the housing, tearing off the cartridge filter housing from its mounting, or simply tearing off the bottom of the housing. All this will ultimately lead to flooding of the premises and headaches for the owner, supplier, seller, and equipment installer. The pressure changes sharply since water is an incompressible liquid. When the water is stopped instantly by quickly closing the valve, the impulse in the closed system must go somewhere - a pressure surge will occur at the valve and a vacuum will form at the other end of the horizontal pipeline [5]. Equally destructive forces develop - the forces of explosion (explosion) and implosion (collapse, explosion inward).

10.2. Using the Comsol simulation software in scientific research

For COMSOL Multiphysics, the Pipe Flow Module is an optional add-on package that models and simulates mass transfer, heat transfer, and fluid flow in pipes and channels (fig. 2). The Water Hammer and Pipe Acoustics interfaces can be utilized to simulate compressible hydraulic transients and acoustic waves, respectively. Problems with temperature, pressure, sound waves, and flow velocity in pipes and channels can all be handled using the Pipe Flow Module. An enormous computational efficiency gain over meshing and computing 3D pipes with finite diameter comes from modeling pipes as curves in 2D or 3D [6].

Piping systems for which the length/diameter ratio is large enough that the flow within each pipe segment can be considered fully developed are suitable for the pipe flow module.

 ISBN 978-3-98924-006-3

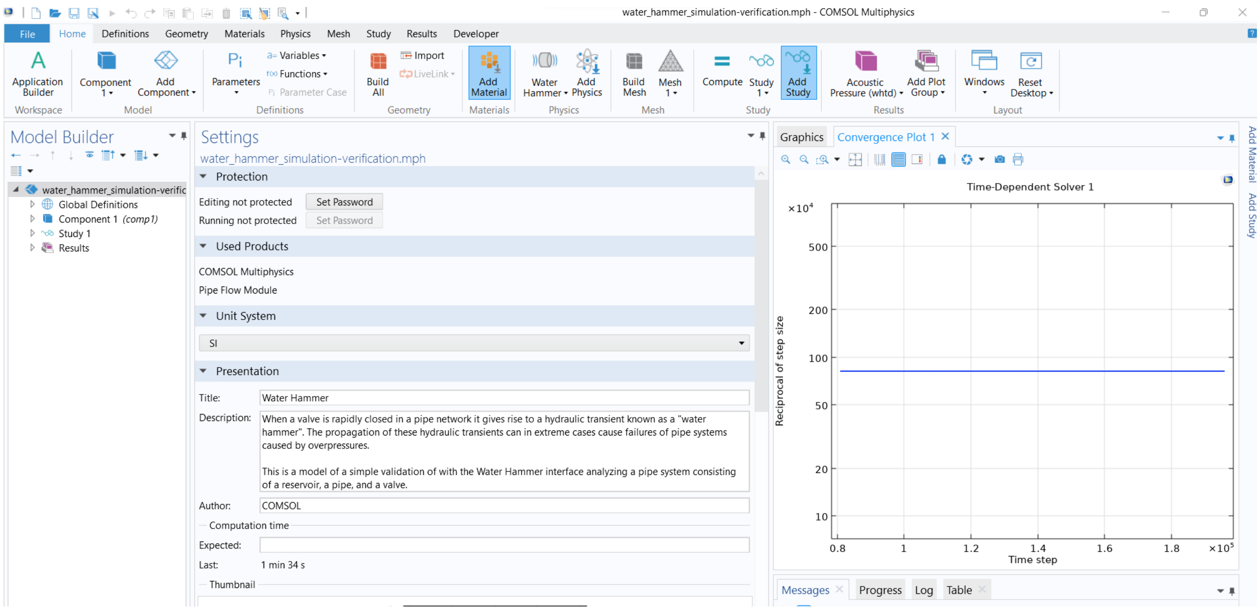

Fig. 2 - User interface of the Comsol simulation software platform

The physical interfaces in the module define the conservation of momentum, energy, and mass of the fluid inside the pipe or channel (fig. 3). The flow rate, pressure, temperature, and concentration fields are modeled as cross-sectional averages, so they vary only along the length of the pipes. Pressure losses along the length of the pipe or in its component are described using friction coefficients. The wide range of built-in expressions for the Darcy and Fanning friction coefficients cover the entire flow regime from laminar to turbulent, Newtonian, and non-Newtonian fluids, various cross-sectional geometries, and a wide range of relative surface roughness values.

In addition to the continuous pressure drop due to friction along pipe sections, the pressure drops due to irreversible losses in components such as bends, constrictions, expansions, tees, y-joints and valves are calculated using an extensive library of industry standard losses coefficients Pumps are also available as flow-inducing devices [7]. The functions of this module are intended for modeling and simulation incompressible and incompressible fluid flows in pipes and channel systems, as well as compressible hydraulic transients and acoustic waves. Typical simulations provide velocity, pressure and temperature fluctuations in pipe and duct systems. Hydraulic transients can also be simulated. This can be the result of a rapid closing of a valve in the pipeline network, which is called water hammer.

The module can be used for the design and optimization of complex cooling systems in turbines, analysis of ventilation systems in buildings, piping systems in the chemical industry, and pipelines in the oil and gas industry, to name just a few applications (fig. 4).

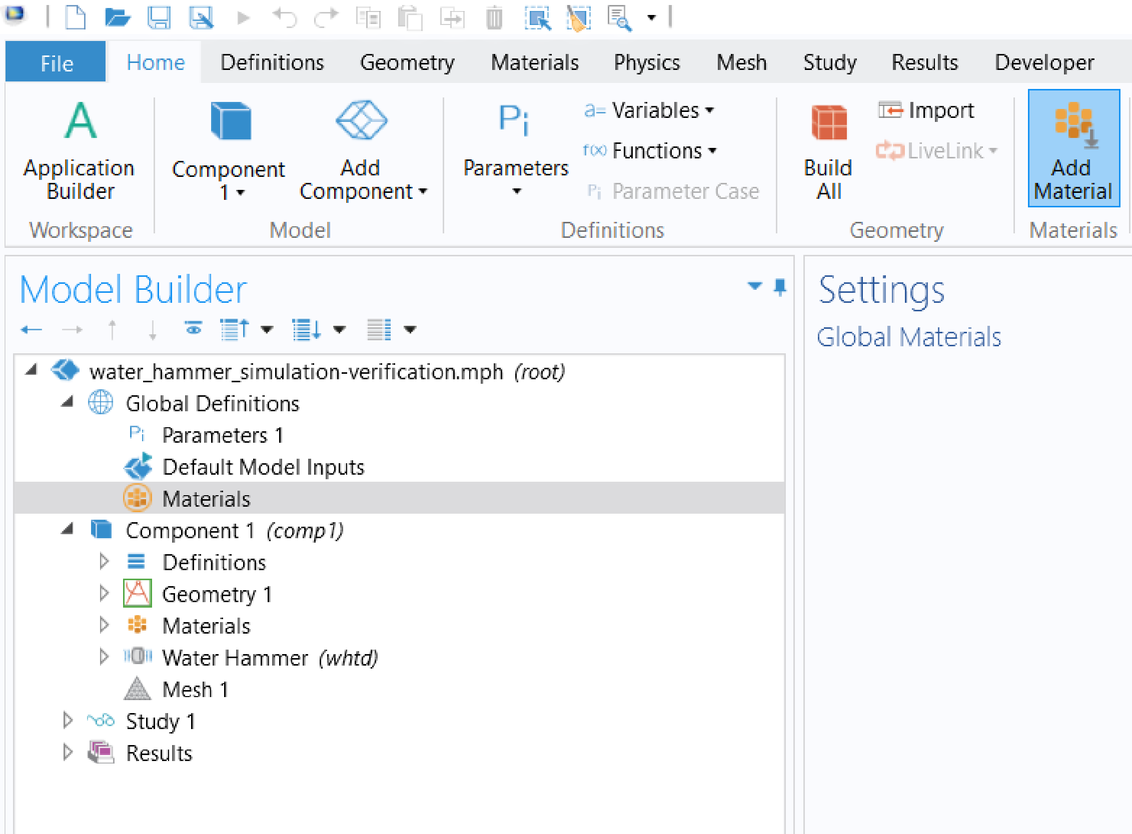

Fig. 3 - Comsol model builder setup configuration

Any devices in which you will experience flow, waves, mass or heat transfer phenomena in narrow pipes or channels are candidates for modeling with the Pipe Flow module.

Classical calculations of pressure drop and mass flow through pipes with elbows, valves, reservoirs, etc. are well suited for the pipe flow module.

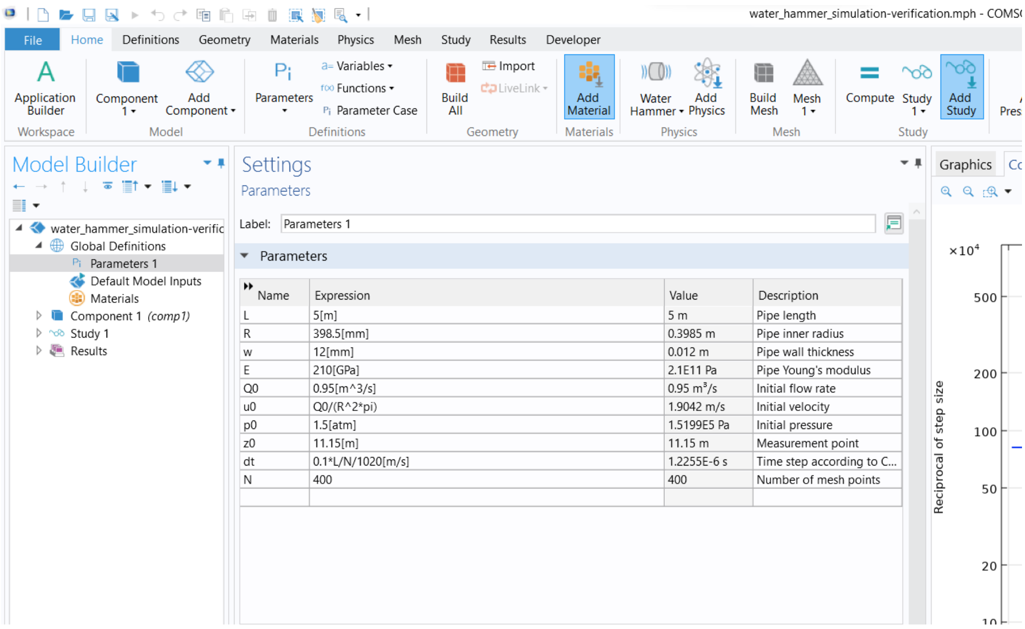

Fig. 4 - Main parameters setup section in Comsol software pacage.

Oil and gas applications can be easily modeled because of COMSOL's strong capabilities in handling so-called nonlinear materials, such as non-Newtonian fluids and materials with highly temperature dependent physical properties (fig. 5). One instance is crude oil pipelines, where the potential for pumping oil is greatly impacted by the effects of viscous heating and temperature-dependent viscosity. A pipe segment can be simply and seamlessly connected to a 3D flow domain using the Pipe Connection multiphysics coupling feature (fig. 6).

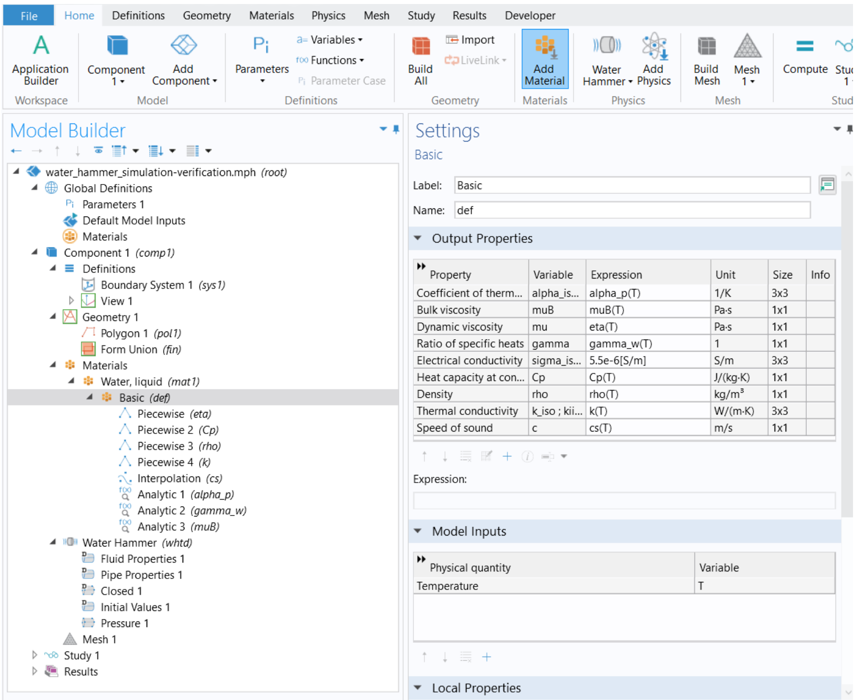

Fig. 5 - Setuping liquid's parameters section for water hamer simulations

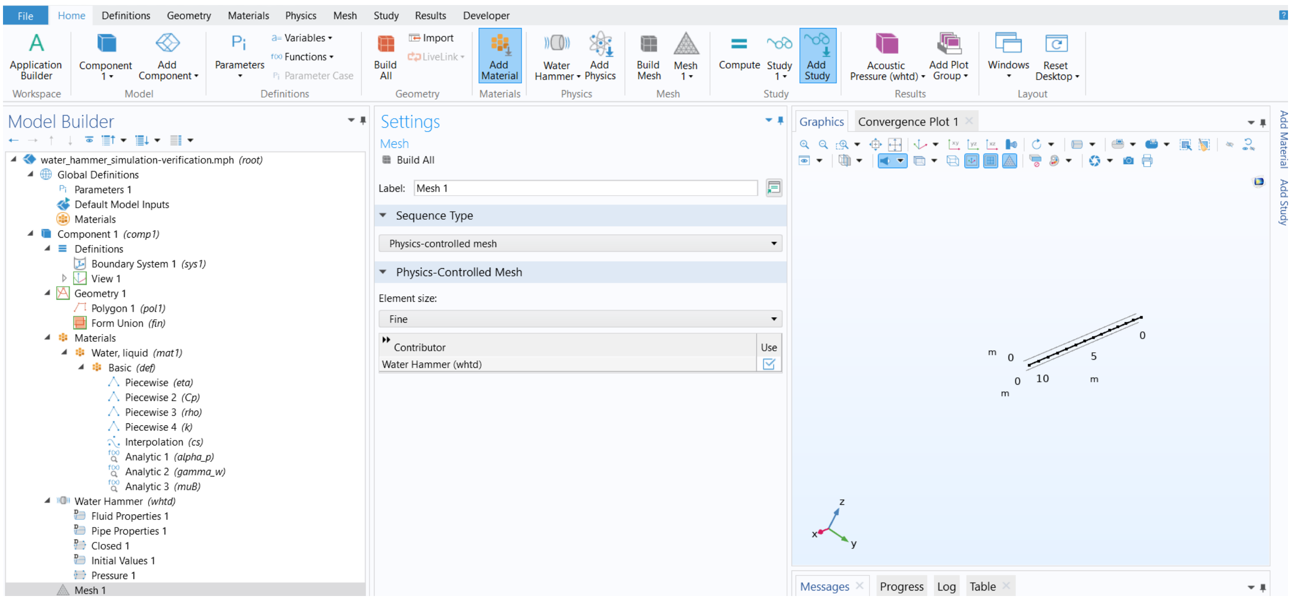

Fig. 6 - Comsol model mesh for water hamer simulation in pipeline

In order to speed up the process, it is possible to adopt the "cover method" of part of the normative documents that require priority implementation.

The Pipe Flow Module is designed to model and simulate acoustic waves, compressible hydraulic transients, and fluid flow in pipes and channel systems. In typical pipe and channel systems, simulations provide the temperature, pressure variation, and velocity. Water hammer, a term used to describe hydraulic transients in a pipe network caused by a quickly closing valve, can also be modelled [8].

The module can be used to design and optimize intricate cooling systems for turbines, building ventilation systems, chemical process pipe systems, and pipelines for the oil and gas sector.

All pipes and channels that can be modeled with the Pipe Flow Module have a pipe length that is sufficiently long to allow the flow inside to be regarded as fully developed. Pumps, T-junctions, bends, valves, contractions, and expansions are among the piping components that are available in the module (fig 7 and fig 8).

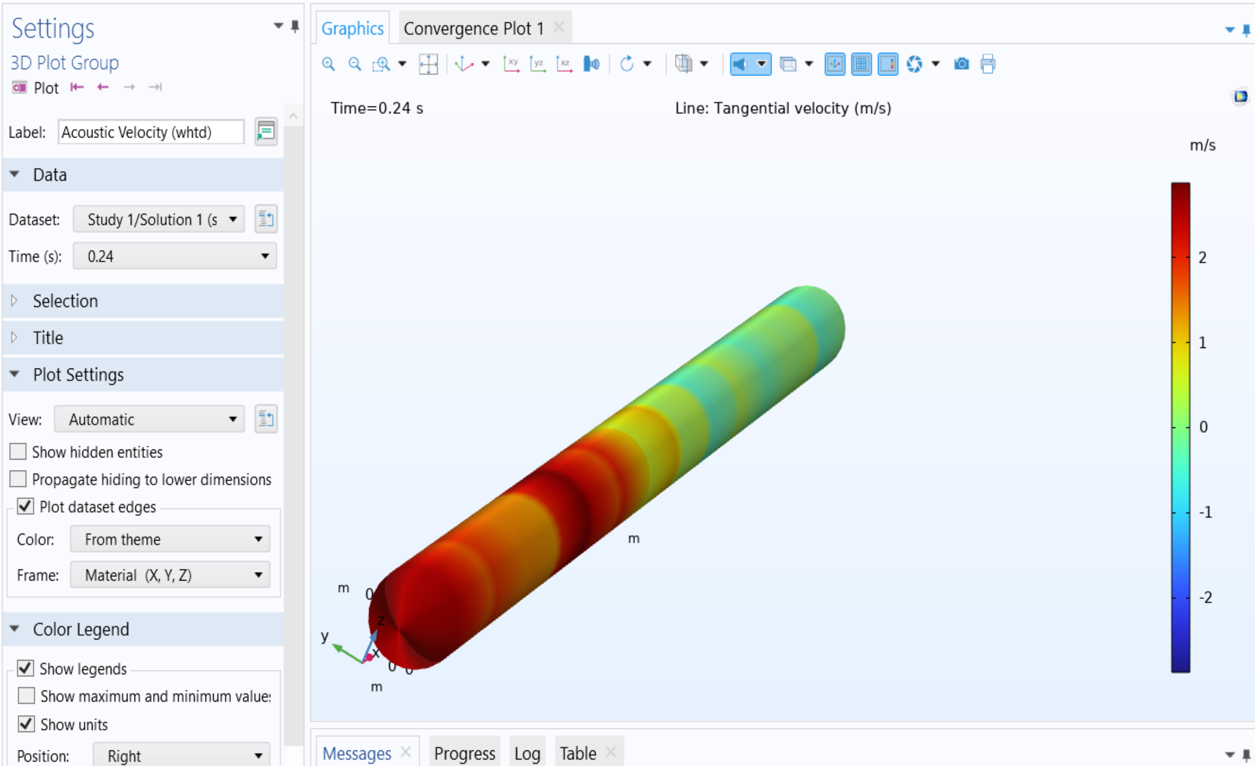

Fig. 7 - Velocity profile for water hammer simulation based onComsol's simulations

The following physics interfaces are included in the module:

• The Pipe Flow interface calculates the pressure and velocity field in systems of isothermal pipes.

• The interface for Heat Transfer in Pipes calculates the energy balance in pipe systems, but it receives the flow field either as a solved field or as a value. Included is wall heat transfer to the surrounding environment.

• The Nonisothermal Pipe Flow interface is a multiphysics interface that solves the flow, pressure, and temperature simultaneously and fully coupled.

• The Transport of Diluted Species in Pipes interface solves a mass balance equation for pipes in order to compute the concentration distribution of a solute in a dilute solution, taking into consideration diffusion, dispersion, convection, and chemical reactions.

• A multiphysics interface, the Reacting Pipe Flow interface solves the simultaneous and fully coupled transport problems of reacting species, pressure, temperature, and flow.

• The Water Hammer interface considers the elastic characteristics of the fluid and pipe wall when resolving fast hydraulic transients in pipe systems.

• Sound waves in flexible pipe systems are modeled by the Pipe Acoustics, Frequency Domain Interface under the presumption of harmonic vibrations.

• Sound waves in flexible pipe systems with arbitrary transient pressure variations are modeled by the Pipe Acoustics, Transient Interface.

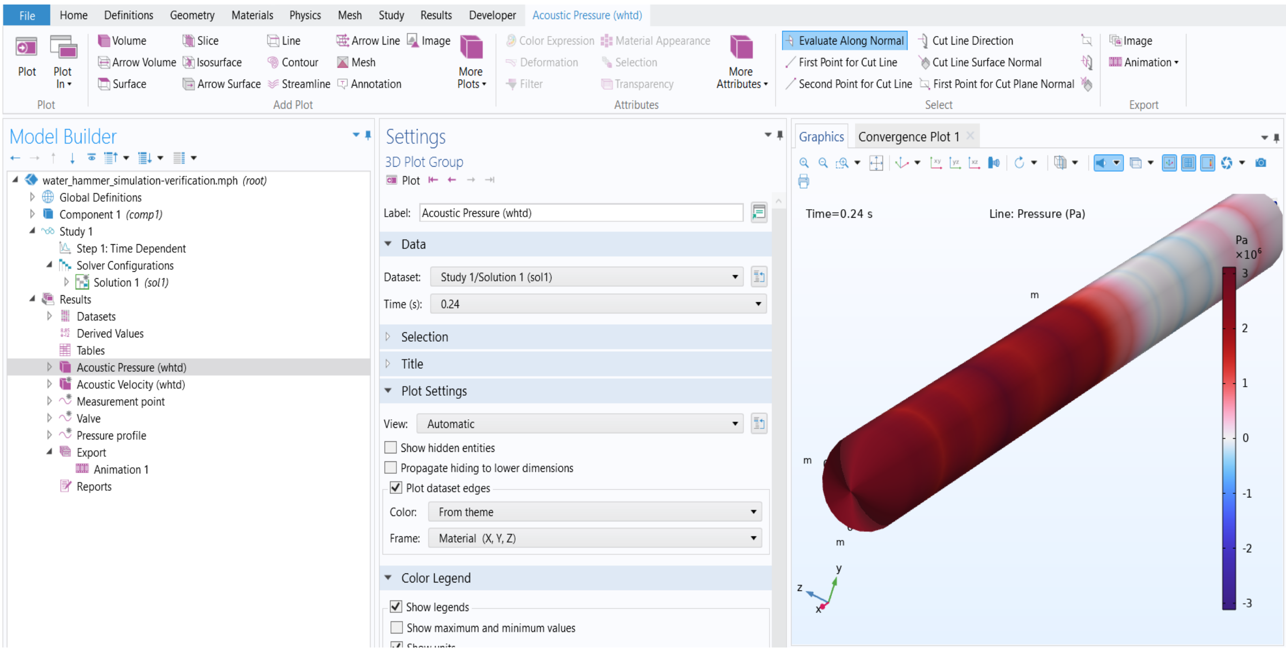

Fig. 8 - Pressure profile for water hammer simulation based on Comsol's simulations

Conclusions

The Pipe Flow Module is utilized to simulate fluid flow, heat and mass transfer, hydraulic transients, and acoustics in pipe and channel networks. It can be seamlessly integrated with any of the other modules in the COMSOL® Product Suite to model the impact of piping on larger entities, such as cooling pipes in engine blocks or connected feeding and product channels. This integration enables efficient utilization of computational resources when modeling processes involving piping networks, while still allowing for a comprehensive representation of process variables within these networks. Pipe flow simulations provide valuable information on velocity, pressure, material concentrations, and temperature distributions along pipes and channels. Additionally, it is capable of simulating acoustic wave propagation and the water hammer effect.

The new design of computing algorithms of the Comsol software has successfully demonstrated a higher output in comparison to the previous model. The velocity and pressure profile simulation results from Figs. 7 and 8 are roughly comparable to the experimental results, respectively. Similar to the Comsol multiphysics simulation, the lack of consideration for frictional loss, head loss, pressure drop, and entry and exit losses limited the simulation's scope (fig 9).

These would make the answer extremely complicated. The pressure distribution obtained in this study is, and it fully resembles the $2.5 x 10^6$ Pa pressure distribution of the CFD analysis by Comsol multiphysics software for simulating real-world processes (fig 10).

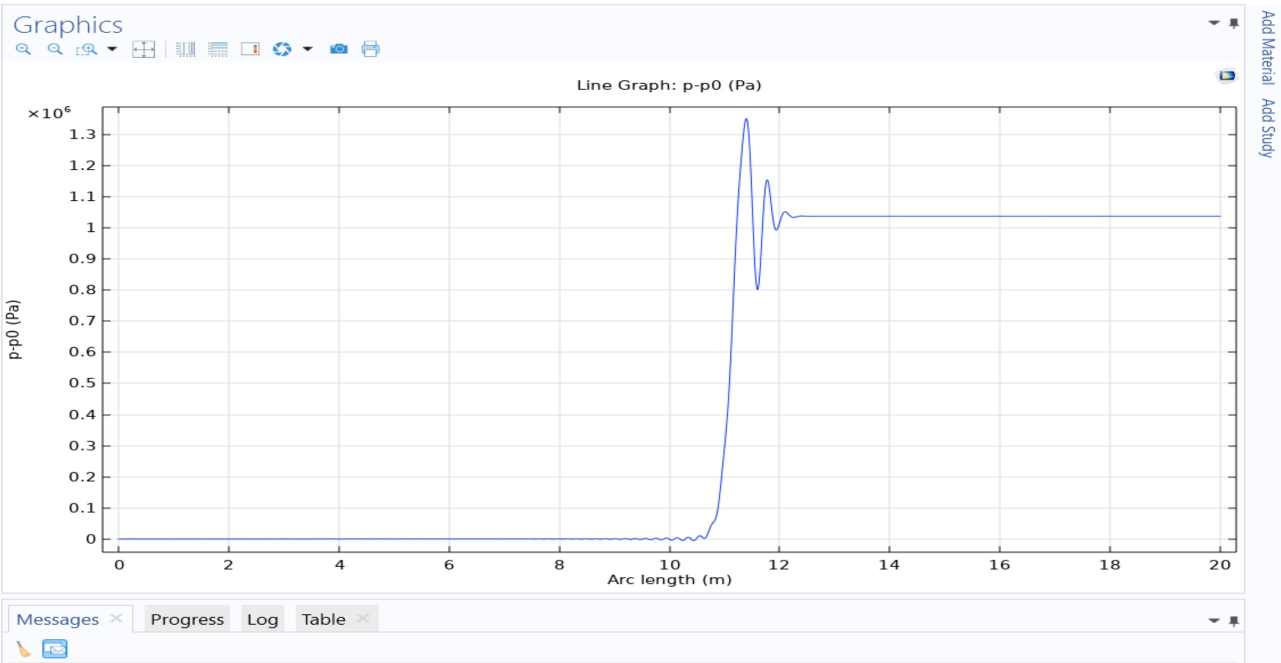

Fig. 9 - Pressure distribution chart for water hammer simulation

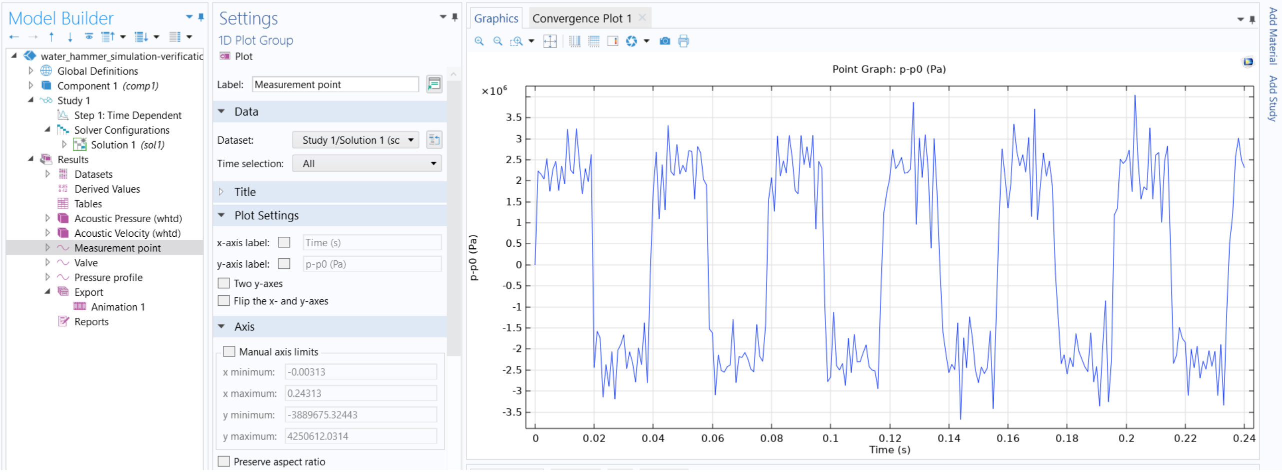

Fig 10 - Pressure distribution chart for water hammer simulation at the initial time period from 0 to 0.25 seconds obtained with Comsol

 ISBN 978-3-98924-006-3