Unsteady Friction Models in 1-dim CFD Simulations for Pump Pipe Interaction with Reciprocating Positive Displacement Pumps

Dipl.-Phys. Marco Klinkigt
Unsteady Friction Model in 1-dim CFD

Simulations for Pump Pipe Interaction with Reciprocating Positive Displacement Pumps

by Dipl.-Phys. Marco Klinkigt
Technical Product Management

Contact:
marco.klinkigt@lewa.de
LEWA GmbH
Ulmer Straße 10, 71229 Leonberg

www.lewa.com
Unsteady Friction Model in 1-dim CFD

Table of contents

1. Introduction .................................................................................................................. 3
2. 1-Dim Computational Fluid Dynamics ........................................................................ 3
3. Friction Models ............................................................................................................. 5
   3.1. Steady Friction Models ........................................................................................... 6
   3.2. Unsteady Friction Models ....................................................................................... 6
4. The Experimental Setup and the Simulation Tool ......................................................... 7
5. Results and Discussion ............................................................................................... 11
6. Conclusions and Outlook ............................................................................................ 13

References ...................................................................................................................... 14
Unsteady Friction Model in 1-dim CFD

1. Introduction

The pump piping system has to be designed carefully if positive reciprocating displacement pumps with high hydraulic power are installed. Improper layout can lead to abnormal high-pressure pulsation, cavitation and fatigue failure of pump, piping or components. Computational simulating tools provide the possibility to predict the pressure and flow pulsation as early as during the design phase. When planning to install reciprocating displacement pumps in a piping system, many companies commission so-called pulsation studies, e.g., according to API 674, in order to appraise the pump-piping installation. Such an evaluation is solely based on computational simulation models. In most of the cases, models of one-dimensional Computational Fluid Dynamics (1-dim CFD) are the best compromise between accuracy and performance. During the derivation of these 1-dim models from the Navier-Stokes equations, the natural friction is lost and has to be added artificially. These friction terms are crucial for the accuracy of the simulation results.

For many years, LEWA has been continuously developing its own software for pump-piping configurations. In recent years, we developed and implemented a powerful one-dimensional fluid dynamics solver. At the same time, a complex experimental set-up was designed and engineered with the aim to evaluate and improve the numerical models used in the 1-dim CFD software. Recently, our focus was on the investigation of different friction models used in 1-dim CFD programs. We examined steady and unsteady friction models and compared the simulation results with measurements received from the experimental setup. In collaboration with our customers, the software tools give us the opportunity to design an efficient pump-piping and dampener device layout for economic and reliable operation.

2. 1-Dim Computational Fluid Dynamics

In numerical simulations of the pump-piping interaction in large and complex piping networks, it is not possible to solve the Navier-Stokes equations, which fully describe the fluid dynamics. There is almost always not enough computer performance available. Moreover, in most of the cases, it is not necessary to know the 'exact' fluid velocity and pressure field within the piping. Very often it is sufficient to have an approximation of the time evolution of the average pressure across the piping cross section area. It needs some assumptions and simplifications in order to
Unsteady Friction Model in 1-dim CFD

obtain equations representing the physical model, which can be solved efficiently on a computer, even for very large and complex piping networks. With the following assumptions, or simplifications, a system of ordinary differential equations, which are used in many 1-dim CFD programs, can be directly derived from the Navier-Stokes equations [3].

The assumptions/simplifications are:

– The flow (and pressure) profile has a rotational symmetry along the pipe axes, when the pipes have circular cross section.
– The velocity of the fluid is negligible against the speed of sound of the fluid. For comparison, typical fluid velocities in the process industries are in the order of 3 m/s. The speed of sound is in the order of 1000 m/s.
– In order to obtain a system of differential equations as simple as possible, the flow (and pressure) gradient rectangular to the axis of rotational symmetry is neglected.
– The properties of the pressure wave propagation are known. The pressure wave propagate with the speed of sound along certain paths in space and time, the so called characteristic lines. In the 1-dim case the pressure wave propagates only in (positive and negative) direction along the pipe.

Given these assumptions/simplifications, we obtain the following well-known ordinary differential equations:

Equation 1 describes a pressure wave propagating in positive direction, and equation 2 describes a pressure wave propagating in negative direction. \( q \) is the volume flow, \( p \) the pressure, \( t \) the time, \( \rho \) the density of the fluid, \( c \) the speed of sound, \( A \) the piping cross section area and \( f \) describes the friction (pressure losses) along the characteristic lines. With only a one-dimensional system, any information of internal friction within the fluid is lost and the
Unsteady Friction Model in 1-dim CFD

friction term $f$ has to be added artificially. Because of its simplicity, equation 1 and 2 can be very efficiently solved on a computer, including for very large piping systems. In order to do this, these equation are integrated along the characteristic lines $C_+$ and $C_-:

\begin{align}
\left( \int_{C_+} dt \left\{ \frac{dq}{dt} + \frac{A}{\rho C} \frac{dp}{dt} \right\} \right) + F &= 0 \\
\left( \int_{C_-} dt \left\{ \frac{dq}{dt} + \frac{A}{\rho C} \frac{dp}{dt} \right\} \right) - F &= 0
\end{align}

The friction term $F$ is now equal to the pressure loss along the characteristic lines $C_+$ and $C_-.$

3. Friction Models

In general, there are two classes of friction models used in 1-dim CFD: models which take the fact into account that the flow velocity profile changes over time, and models which do not. Of course, friction models which take the change of the flow velocity profile over time into account are difficult with respect to modeling, implementation and computational performance. The main difference is that the pressure loss not only depends on the actual fluid velocity, it also depends on its history, its evolution over time.

In many applications, the volume flow does not change or changes only very slowly over time, e.g. in systems with centrifugal pumps. However, in the case of reciprocating displacement pumps\(^1\), the volume flow typically changes very strongly and very fast. Hence, there will NEVER be a fully developed flow profile within the pipe. Implementing a friction model which assumes steady flow will lead to qualitatively and quantitatively wrong simulation results in this case. Measurements in systems with strongly fluctuating fluid velocities show in many cases that 1-

\[1\] Reciprocating positive displacement pumps are machines with eccentric-cam-driven pistons. Since flow is a product of piston displacement, the fluid accelerates and decelerates at each stroke of the piston(s). The result is a pulsating/oscillating flow regimen, as opposed to the nearly constant flow regimen of a centrifugal pump. Unlike the centrifugal pump, therefore, considerably more attention to detail is essential when designing the piping system. From variations in fluid velocity as the result of corresponding variations in pressure, the relevant amplitudes can result in cavitation, overload, noise, and / or strong piping vibration that can lead to fatigue failures in both the pump and the piping system [2].
Unsteady Friction Model in 1-dim CFD

dim CFD simulations using steady friction models overestimate the pressure fluctuation. During the design phase, the overestimation sometimes leads to the selection of uneconomically large dampener devices, e.g. resonators or to unnecessary orifice arrangements within the piping.

3.1. Steady Friction Models

In a Newtonian fluid, the friction is proportional to the gradient of the fluid velocity rectangular to the direction of flow. The easiest approach to model the friction function $F$ in equation 2 and 1 is to assume a steady flow. In this case, the flow does not depend on time, and the friction function $F$ in equation 2 and 1 is well known, because the fluid velocity profile across the pipe cross section is known. In a system with steady flow or with a flow which changes only very slowly over time, $F$ is a simple function and depends only on the viscosity, the roughness of the piping walls, and the fluid velocity. $F$ can be written in the following form:

$$F_{\text{steady}} = \frac{\rho}{2} \cdot v^2$$

Where $\rho$ stands for the density of the fluid, $v$ for the fluid velocity, $\zeta$ is the so called friction coefficient, depending on the cross section area $A$ of the piping, the Reynolds number $Re$, the fluid viscosity $\eta$ and the roughness of the piping walls $\chi$.

3.2. Unsteady Friction Models

In the case of ('fast' running) reciprocating displacement pumps, the flow velocity fluctuates strongly on a short timescale. In general, if the flow velocity changes fast, the corresponding flow profile which is directly associated with the friction is no longer a simple function of the fluid velocity. The mathematical description of the friction term in the piping is now much more complicated. In 1968, Zielke [4],[5] made a suggestion in which:

$$F_{\text{unsteady}}^{\text{app}}(t) \propto \int_0^t \frac{\partial \nu}{\partial t}(\alpha) \cdot \nu(t - \alpha) \, d\alpha.$$
Unsteady Friction Model in 1-dim CFD

Where \( \dot{v} \) is the instantaneous mean flow velocity, \( t \) the time, \( w(t) \) is a weighting function and \( \alpha \) the time used in the convolution integral. Using such an expression in the simulation model involved a numerical integration of the previous velocity history at each time step and each grid point and is, therefore, very inefficient from a computational point of view. Trikha (1975) [4] derived an approximation of the Zielke weighting function \( w(t) \). With this approximation, the friction function has the following mathematical structure:

\[
F_{\text{unsteady}}^{\text{app}} = \beta_1(\eta, A) \cdot q(t) + \beta_2(\eta, A) \cdot \sum_{i=1}^{n} y_i(t) \cdot y_i(t - \Delta t) \cdot v(t - \Delta t)
\]

Where \( \beta_1 \) and \( \beta_2 \) are factors that depend on the pipe cross section area \( A \) and the viscosity \( \eta \). \( q \) is the volume flow of the fluid. The function \( y_i \) depends on the value of the flow velocity at the current time and the value of \( y_i \) and \( v \) of the last time step. The improvement concerning the computational performance is that only information of the last time step is required, and not information of the complete previous flow velocity history.

4. The Experimental Setup and the Simulation Tool

In order to evaluate the numerical model implemented in the 1-dim CFD simulation tool, an experimental setup was engineered (Figure 1). The piping length between manifold and vessel is adaptable from 20 [m] up to 80 [m]. The inner pipe diameter on the discharge side is 9 [mm]. Installed on the experimental setup is a reciprocating metering pump, type LDC-M910S-Size \(^2\) in triplex configuration with variable stroke length, (0 [mm] - 15 [mm]) at each pump head. The metering pump can operate in single, duplex and triplex configuration. The maximum pump speed is 420 [spm]. The discharge pressure is adjustable in the range between 1 [bar] and 146 [bar] absolute. The test bench allows to measure the pressure at many different positions during operation. The pipe routing can easily be modified by opening and closing valves. Resonators and dampener devices can be installed at different positions. The test fluid is water.

Figure 2 shows a model of the test bench. The 1-dim CFD solver which is used to examine the different friction models, was developed by LEWA GmbH and uses the method of

\(^2\) manufacturer: LEWA
Unsteady Friction Model in 1-dim CFD

characteristics, which is briefly discussed in the section '1-Dim Computational Fluid Dynamics'. Implemented are two friction models, steady and unsteady (for further explanation, see the section 'Friction Models'). The steady friction model was checked against commercial third-party software, also using the method of characteristics. We proved that the results calculated with LEWA software and the results calculated with the third-party software are the same. The module with the unsteady friction model could not be tested against a third-party software, because we have no knowledge of available 1-dim CFD programs in which the unsteady friction model, discussed in the preview section, is implemented. Besides the piping friction, the software also takes care of the acoustical effect caused by bends, abrupt expansions, or contraction and various armatures. Also included are models for air vessel, bladder type dampener and resonators. The software has an interface to a third-party software, providing the possibility to simulate the mechanical response of the piping, based on the 1-dim CFD pressure pulsation results.
Unsteady Friction Model in 1-dim CFD

Fig. 1: Experimental setup for evaluating the simulation software. The installation features a reciprocating metering pump, type LDC-M910S-Size 21 (manufacturer: LEWA), in triplex configuration with variable stroke length at each pump head. The pump can operate in single, duplex and triplex configuration. The maximum pump speed is 420 [spm].
Unsteady Friction Model in 1-dim CFD

Fig. 2: Simulation model (discharge side) of the test bench.
Unsteady Friction Model in 1-dim CFD

5. Results and Discussion

We performed various measurements with the experimental setup, which is described in the section above. Many different operating modes, with different piping routings or, lengths, different stroke lengths, different pump speeds, in single, duplex and triplex configurations were examined during an extensive simulation model evaluation process. The comparison of the simulation results with the results obtained from the corresponding experiments clearly shows, that the unsteady friction model demonstrates the qualitatively and quantitatively better agreement with the results of the measurements. Exemplary results are plotted in Figure 3, 4 and 5. Concerning the deviation of the measurement with the simulation results, two things are noticeable.

Using the steady friction model, the value of the peak-to-peak pressure pulsation \( p_{\text{max}}[t] - p_{\text{min}}[t] \) is too large in almost all considered cases. In some cases, a deviation of more than two hundred percent was observed. The results from the calculation with the unsteady friction model shows a very good fit with the values from the measurement. Figure 3, shows an exemplary result. In this example, the pump operates in duplex operation with 60 strokes per minute [spm].

Another significant difference between the simulation results obtained by the calculation runs with different friction models is the decay behavior of the initial pressure spike (refer to Figure 3). The initial valve opening pressure spike arises from the finite compressibility of the fluid. Due to the compressibility, the valves have a delayed opening. Once the valves open, the plunger velocity is not zero and the coupling between the fluid in the pipe and the fluid in the pump head can be very violent, leading to the so-called valve opening pressure spikes. The measurement shows a very fast decay of the valve opening pressure spike. The simulation with the unsteady friction model reproduces almost the same decay behavior (Figure 3). The result obtained with the steady friction model shows a decay behavior, which is much too slow. At the beginning of the discharge stroke of the second pump head, the measurement shows a pressure pulsation with a peak-to-peak value on the order of 0.1 [bar]. The result from the calculation with the steady friction model shows a value that is still on the order of 3 [bar] at this point in time.

The overestimation of the peak-to-peak pressure value and the wrong decay behavior of the initial valve opening pressure spike, using the steady friction models, was observed in many test
Unsteady Friction Model in 1-dim CFD

procedure measurements. These measurements take place at LEWA on a permanent basis for various pump sizes and pipe dimensions.

Fig. 3: Comparison of the simulation results with the measurement. Left: The black solid line shows the result of the measurement, the grey solid line the simulation result with the steady friction model. Right: The black solid line shows the result of the measurement, the red solid line the simulation result with the unsteady friction model. The pump runs with 60 [spm] in duplex configuration.

Fig. 4: Comparison of the simulation results with the measurement. Left: The black solid line shows the result of the measurement, the grey solid line the simulation result with the steady friction model. Right: The black solid line shows the result of the measurement, the red solid line the simulation result with the unsteady friction model. The pump runs with 200 [spm] in single head configuration.
Unsteady Friction Model in 1-dim CFD

Fig. 5: Comparison of the simulation results with the measurement. Left: The black solid line shows the result of the measurement, the grey solid line the simulation result with the steady friction model. Right: The black solid line shows the result of the measurement, the red solid line the simulation result with the unsteady friction model. The pump runs with 50 [spm] in triplex configuration.

6. Conclusions and Outlook

We showed that using unsteady friction models in 1-dim CFD provide very good agreement with the corresponding measurement data. The peak-to-peak pressure pulsation as well as the decay behavior of the initial valve opening pressure spike fits the experimental data. The simulations with the steady friction models show significant deviations in both, the value of the peak-to-peak pressure pulsation and the decay of the initial valve opening pressure spike. Whereas the decay behavior is not strong enough, the peak-to-peak pressure values are overestimated.

A pulsation study for a pump piping system with reciprocating pumps usually provides the following as outcome: dampener sizes, values for the minimum required suction pressure, and further arrangements in order to reduce pressure pulsations below the required limit. Unsteady friction models in 1-dim CFD help to avoid unnecessary, uneconomic or, in the worst case, useless measures to reduce pressure pulsation.

In the near future, our focus of investigations will be on the evaluation of dampening device models, such as bladder type, air vessel and resonators. In a second step, the investigation will be extended to the mechanical response simulation of the piping elements and armatures, based on the pressure pulsation results generated with 1-dim CFD.
Unsteady Friction Model in 1-dim CFD

References


