HYDRAULIC MODELING AND SIMULATION OF PUMPING SYSTEMS

by
Augusto Garcia-Hernandez
Senior Research Engineer

Melissa Wilcox
Research Engineer
and
Thomas Moore
Senior Research Engineer
Southwest Research Institute
San Antonio, Texas

Augusto Garcia-Hernandez is a Senior Research Engineer at Southwest Research Institute, in San Antonio, Texas. His professional experiences include various experimental works, sales, technical support, drilling research assistant, and drilling laboratory teaching assistant. His Master's thesis focused on cuttings transport velocity in horizontal and highly-inclined wells. In addition to his graduate work, he has been involved in research on double-piston pumps (experimental characterization and evaluation) and progressive cavity pumps (modeling development), as well as drilling fluids applications. Mr. Garcia has publications in areas such as cuttings transport in horizontal and deviated wells, gas and liquid pipeline transport, and centrifugal compressor surge analysis.

Mr. Garcia-Hernandez holds a B.S. degree (Mechanical Engineering) from Central University of Venezuela and an M.S. degree (Petroleum Engineering) from the University of Tulsa.

Melissa Wilcox is a Research Engineer at Southwest Research Institute, in San Antonio, Texas. Her background includes work related to analysis and testing of gas turbines, compressors, turbines, compressors, and pipeline systems. She has conducted various modeling efforts of pumping systems including liquids pipeline systems, determining NPSH of a pump/piping system, and pump operation. Recently she has been involved in the design of a pump test facility for pipeline pumps.

Ms. Wilcox has a B.S. degree (Mechanical Engineering) from Texas A&M University and graduate work at Georgia Institute of Technology.

Thomas Moore is a Senior Research Engineer at Southwest Research Institute, in San Antonio, Texas. He is actively involved in a variety of projects throughout the Institute. Some of the project work includes hydraulic simulations, risk and reliability analysis, life cycle cost analysis, energetic containment analysis, finite element analysis, laser electro-optics, and fluid flow studies.

Mr. Moore received a B.S. degree (Physics) and an M.S. degree (Visualization Science) from Texas A&M University. He participates in a number of professional societies and is a member of the American Physical Society, Optical Society of America, and the American Institute of Aeronautics and Astronautics.

ABSTRACT

New designs or modifications of existing pump facilities are required frequently in many industries due to new operating conditions, increase in product demand, pump-pipeline interaction issues, pulsations problems, net positive suction head (NPSH) requirements, and so forth. In many cases, hydraulic calculations are carried out by hand, within spreadsheets or using hydraulic simulation tools. Computational tools have become very popular and practical for simulating the interaction of the different pumping facilities within the pipeline system as well as defining piping layout and different configurations. However, there is not a well-defined methodology for approaching the diverse types of issues encountered within the industry. A general approach for analyzing some of the typical cases observed in many pumping facilities through hydraulic modeling is presented in this tutorial.

The general hydraulic modeling methodology presented will cover various aspects of analyzing a pumping system such as: initial approach, data requirements, hydraulic pipeline modeling, computational fluid dynamics (CFD) modeling, and model validation. The methodology will start from a basic definition of the problem and tools used for this type of analysis. This will assist in defining an optimal path to model and simulate the system. The tutorial will conclude with basic case studies of the pumping systems that will illustrate the application of the methodology, its basic assumptions, results, and limitations.

INTRODUCTION

Pump system design is encountered when new pumps are being installed or an existing system is being retrofitted, expanded, or modified. The main focus of the design process is fulfilling the operating requirements for the application, but this involves consideration of many other elements of the pump system besides the pump itself. Operational considerations include net positive suction head (NPSH) requirements, pump power, required head or total developed head (TDH), piping layout, possible pump interactions, pipeline acoustics, and flow pulsations. Some of the hardware that is required for the operation of the pump systems is valves, control systems, regulators, piping, relief valves, vessels, and drivers (motor, turbine, and engine). Each of these components must be selected and operated on the basis of meeting cost, reliability, availability, and maintenance requirements in addition to meeting the basic technical requirements. One factor that often drives the final design is the technical-economical relationship. This is usually assessed with a life cycle cost analysis. Past experience can contribute to this study.

There are many challenges that can arise during design, installation, and operation of the pump system. Past experience has shown issues during operation can be minimized if more effort is invested up front in the design and installation. Some minor issues that often occur during start-up and operation are leaks, problems with the logic and communication of the control system, and minor vibrations due to unbalance and misalignment. Some major issues that should be addressed during the design phase are cavitation, piping pulsations, large vibrations, unacceptable pump supports, inadequate supervisory control and data acquisition (SCADA) and programmable logic controls, and insufficient power and heat capacities. Many of these issues can be quantified and successfully addressed with the use of hydraulic modeling and simulation.

The design or analysis process for each pump and application has unique concerns, issues, and design features. However, general design methods, analysis, and tools can be applied to aid in the engineering process. Some tools that can be used are hydraulic modeling, computational fluid dynamics (CFD), and process simulators. Pipeline and process simulators are very beneficial for developing acceptable engineering solutions to a variety of challenges since they are able to model the real system, have dynamic capabilities, and are interactive, flexible, and multifunctional. Typical applications include calculations of pumping-storage facilities steady conditions, transient analysis of pipeline-pump interaction, water hammer effect, start-up and shutdown sequences, and modeling of transmission and interconnecting pipelines. CFD is another powerful tool that can be used to understand and solve many of the hydraulic problems encountered in a pumping facility. Predictions of cavitation, vortex formation, velocity profiles in elbows and suction nozzles, study of impeller modifications and efficiency improvements are some of the typical applications of a CFD analysis in a pumping system.

This paper reviews the tools available for pump system design and analysis. The discussion focuses on the important parameters that are used for an analysis. Lastly, case studies are presented for hydraulic and CFD models.

FLUID, EOS, AND BOUNDARY CONDITIONS DEFINITION

Fluid properties and the proper use of the equation of state (EOS) should be considered an important part of the model. Small mismatching can significantly affect the analysis results. Liquid viscosity is critical for fluids that have a high temperature dependence and when considering friction losses. In addition, density values should be monitored carefully since high elevation changes have direct effects on the system due to gravitational losses. Another important consideration is the effect of the bulk modulus or compressibility value when transient conditions are analyzed. The speed of sound calculations are dependent on these values.

Fluid Properties Needed for a Typical Analysis

It would be ideal to have a full fluid composition for an analysis. However, this is not available in all cases. With the full composition, a variety of fluid properties such as density, viscosity, bulk modulus, vapor pressure, heat and thermal conductivity, and diffusion coefficient can be calculated at different pressure and temperature values with an EOS solver.

As a minimum, the liquid model should have density, viscosity, vapor pressure and bulk modulus for the fluid of interest. A general understanding of the fluid and its behavior in the hydraulic system is required to make the proper assumptions in order to generate acceptable inputs for the fluid properties. In addition, an understanding of the modeled system will provide valuable insight for determining the necessary fluid data and the uncertainty associated with that data.

Boundary Conditions Required

To have an accurate or representative analysis, the boundary conditions of the model should be set to represent the actual operating conditions. The first thing to consider is the effect and importance of defining the proper conditions of the system. Flow, pressure, temperature, and speed are common boundary conditions for a pumping system. The best approach is to define the boundary condition as they will be in the real system. For example, the flow through a pipeline system is controlled by a maximum pressure at the pumping station and a back-up pressure at the receiving facility to avoid slack conditions. Therefore, a pressure condition would be the boundary condition at the end of the pipe while flow will be adjusted at the pumping facility to match actual operating conditions. It should be noted that a pressure and flow condition is necessary in order to calculate a unique solution.

PUMPING SYSTEM

The primary considerations in the design of pump stations are hydraulic requirements, possible system constraints, applicable international and local standards, safety, reliability, availability, flexibility for future expansions, environmental impact, and cost. Pumping systems are used in a wide variety of applications. However, the main concept is the same, transport fluid from one location to another in the safest, fastest and most efficient manner. Pumping systems usually handle variable heads and flow rates. They are used in several applications that include hydrocarbon fluids, water, refrigerants, food, industrial application fluids, and so on. These applications cover multiple products with a wide range of viscosities and fluid properties. The study and analysis of pumping systems involves a relationship between the application, type of pump equipment and its driver. Pumps are classified based on the method used to move the fluid. The two major groups include positive displacement and rotordynamic pumps. The driver selection will depend on the type of facility, space and fuel available, required reliability, equipment compatibility, and applicable standards. Typical drivers used include electric motor, diesel engines, and gas turbines.

Input Parameters Required for Different Types of Analysis

Pumping systems require special attention since the safety considerations and risk are much higher than in other process systems. Different scenarios can be observed in the same application and facility. It is important to understand the primary objective of the study before proceeding with a detailed analysis. Typical analyses include design of new installations, modifications or upgrades of existing facilities due to an increase of capacity, root cause failure analysis, flow and power optimization, and definition of new operating philosophies or process controls. In general, an analysis will start with defining the hydraulics requirements, type of equipment used or to be installed, and primary objective and variables to be studied.

For example, in a new design or modification with centrifugal pumps for the petroleum, petrochemical and natural gas industries a good guideline is the ISO 13709 Standard (2007). This document covers hydraulic calculations, pump types, required seals materials and geometry, bearings, lubrications, drivers' selection, accessories, instrumentations, testing, inspections, and many other important subjects.

The computational models should include the following characteristics and elements:

1. Main process boundary conditions, constant flow, or pressure at the inlet and outlet of the pumping systems

2. Tanks or reservoirs considering their actual volume and geometry

3. Pump's performance characteristics and operating conditions

4. Ambient temperature

5. Recycle loop for each pump and a bypass valve for each pipeline station

6. Fluid properties at various temperatures such as viscosity, density, vapor pressure, bulk modulus, etc.

7. EOS definition, for example the slightly compressible liquid (SCL) is usually used for batched tracking of fluids, blending, and definition of several liquids with their respective percent composition.

8. Piping layout, geometry, and elevation profiles.

9. Surrounding average ambient and soil temperatures based on the site ambient conditions

10. Heat transfer coefficients for the pipelines, coatings, and soil.

11. Valves are part of the model to allow the start-up and shutdown of the pumping stations, process control and isolation of specific pipeline branches or sections.

12. Heaters, coolers and separators are usually simulated as pipe volume including their expected pressure drop and temperature load.

Operation of Pump in Simulator

Pump equipment is incorporated in the computational models with as much detail as possible to produce accurate results. Pump performance characteristics should include head versus flow curves, power, NPSHR and efficiency at various speed conditions as well as inlet and outlet loss. It is essential to know what fluid will be pumped and its properties. Performance curves are usually generated for water; therefore, they should be corrected for high viscosity values and non-Newtonian fluids. Other important parameters to be included in a pump model are the moment of inertia, number of stages, minimum and maximum flow at rated speed, and average starting torque. Best efficiency point and rated conditions should be specified as well.

Hydraulic pump simulations resemble real operation conditions accurately. The computer model calculates all the hydraulic behavior based on the performance curves and data provided. These calculations are linked to the hydraulics of the simulated system. Operating and boundary conditions such as flow or back pressure regulation can be modified interactively while conducting dynamic simulations. In addition, start-up and shutdown condition can be simulated to calculated effective start-up torque and flow transients in the system. Actual flow, head, pump speed, efficiency, NPSHR, NPSHA, suction and discharge pressures and temperatures, are calculated by the software at every time step and reported back to the user for analysis. Therefore, steady-state and transient conditions can be analyzed in depth without sacrificing accuracy. The display of the results usually includes a performance map and operating conditions as shown in Figure 1. A detailed report of almost any parameter can be obtained in a table or chart form.

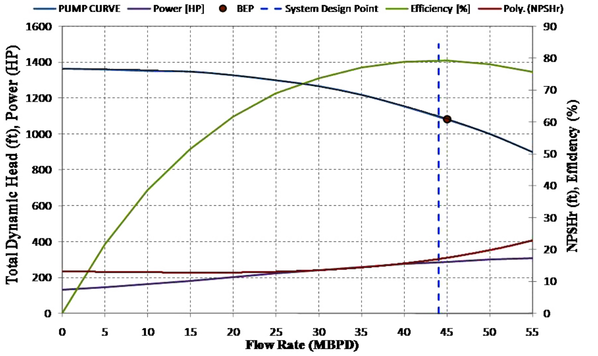


Figure 1. Example of a Pump Performance Curve.

Tuning the pump model is necessary to obtain accurate results. The tuning process depends on the availability of field or experimental data for comparison and refining. The model is initially built with all the theoretical data available and then it is run at known field conditions for comparison. Usually, parameters such as inlet and outlet loss coefficients, shaft friction losses and system hydraulic are adjusted to tune a pump model. However, it is difficult to obtain a complete match between model results and a system's real data. Therefore, at the end of this process a model uncertainty is calculated for the most important parameters such as head, flow, temperature, power, speed.

Transient events are computationally demanding and they require well-defined sequences of events to be computed accurately. In addition, fluid transients observed in the pipeline system will affect the behavior of the pump. In some cases, small changes in the system are handled without any issues by the pumping station, while drastic changes in the pipeline system could generate a trip condition such as high pressure or low flow for the pumping system. An example of a critical transient event is presented in Figure 2. This event represents an unexpected valve closure downstream in the piping system that generated a shutdown of an entire pumping station due to high pressure.

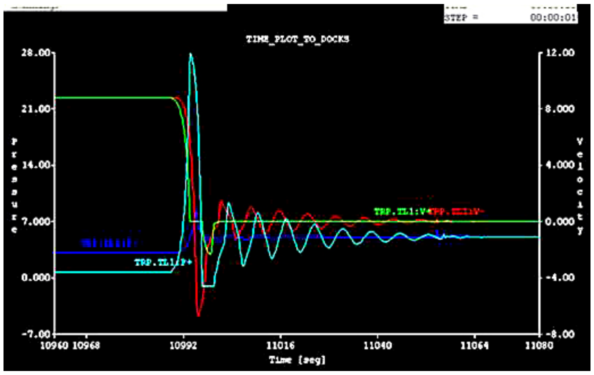


Figure 2. Example of a Transient Event that Affected a Pumping System.

Pipeline System

The pipeline system requires special attention when modeling, since it involves a vast amount of information and assumptions. Preliminary analysis of the pipeline systems is commonly performed with hydraulic simulations to evaluate parameters such as pipe size, maximum capacity, maximum allowable operating pressure (MAOP), pumping station spacing and configurations and pressure ratios.

Optimization, economical evaluations and definition of operating philosophies are usually aided by pipeline modeling. In several applications, online models are used to help with the capacity forecast and operational decisions on a daily basis.

Input Parameters Required for Analysis of Piping Systems

In general, a detailed analysis of a pipeline process entails several parameters such as maximum and minimum flows, pipeline routing, load factor, storage capacity in the stations, pipe size, pump stations spacing, environmental/socioeconomic factors, pump drivers and power available, fuel or electricity cost, and heating or cooling requirements. This is a complex and iterative process that involves a detailed analysis of the different alternatives. It is usually complemented with a sensitivity analysis of the most important techno-economical variables, such as nominal flow to be transported, actual total cost (operational + maintenance + expansion) and forecasted values, expected revenues, amortization, return on investment (ROI) period and other important indicators.

In addition, risk analysis tools and reliability assessments are combined with pipeline simulations to aid in new design layouts, modifications or optimization of existing systems. For example, pumping station spacing and power can be calculated using a pipeline modeling tool while a reliability analysis of the stations can be obtained through a Monte-Carlo simulation. Therefore an optimization of power usage, ROI and risk can be accomplished by combining those simulations tools.

Tuning of Model Based on Experimental Data

Currently, computational models are very common and useful for providing quick, reliable, and cost-effective solutions to real problems. In general, pipeline models include a lot of detailed and specific information of the real system being modeled. Therefore, it is important that these data be accurate in order to ensure the predictive capability of the computational model. However, computational simulations have some level of uncertainty and inaccuracy due to the use of simplifications, assumptions, and numeric calculations. Therefore, model developers have to adjust, refine, and validate their models to simulate the real process and events more accurately. Models are validated by comparing the simulation results to known steady-state and transient parameters at various other operating points.

The model refining is a trial and error process that allows improving the accuracy of the model by comparing its predictions against known conditions. In a pipeline system standard friction factors, head transfer coefficients, pipeline efficiency, soil thermal conductivity and properties are estimated initially based on the data available. However, those parameters can be adjusted to refine the model results. An initial set of data is utilized to tune the model while a second data point is used to validate the tuned model. The model refining should be conducted with extreme operating values in the operating range to be studied. In addition, the refining process may affect more than one variable or parameter in each iteration. For example, a real temperature profile is used to tune the thermal properties of the soil in a heavy oil pipeline transmission line. This temperature adjustment will influence the calculations of the fluid properties along the system affecting the pressure loss predictions. Therefore, a new iteration will be required since these two parameters are dependent on one another. In this specific case friction affects the temperature significantly and vice-versa.

Transient Events

Many times during the design process, the main operating conditions modeled for a system are the conditions at steady-state. Real systems do not operate at this steady-state 100 percent of the time. There are time varying or transient events that occur. Some examples of these are start-up, shut down, liquid batching changing process, process upsets, etc. During these events the pressure and flow are changing in the pipeline to match the changes in the process.

Hydraulic simulators can be used to model such events. The pump curves, start-up speed curves, valving procedures, emergency shutdowns, etc., can be modeled. Individual pump elements are included in the model, which contain all the features of a real pump. The pump curves are used to define how the pump will operate and react to a changing system. The speed of the pump can be controlled to match the expected start-up procedures. Valves can be controlled, monitored, and actuated to match the valving process that would occur during a batching change operation.

While these transient events are being simulated, time and spatial plots can be generated for pressure, flow, temperature, etc., to monitor how the liquid is changing with the events. These simulations could be used to determine how the system will react during an upset, identify if the pressure will go above the MAOP, or system sequences used are timed correctly for the operation.

Fluid Hydraulics

There are many components on the piping system that create varying pressure drop depending on the fluid traveling through the pipe as well as the velocity of the fluid. It is important that these components are modeled correctly so the pressure changes across the pipelines will be accurate.

Pipes are modeled based on their length, diameter, and pipe resistance. The pipe resistance can be input into the model as a surface roughness or friction factor. The friction factor is dependent on the Reynolds number of the velocity of the flow, which is not always constant. If possible, the surface roughness should be used for the pipe resistance. The modeling program can then calculate the appropriate friction at different operating flow or fluid velocities. Typical surface roughness for steel pipes range from 0.00009 to 0.0002 ft.

Each valve in the system will have a pressure drop across it. This pressure differential depends on the type and size of the valve and the velocity of flow through the valve. Valves are defined in hydraulic models by valve coefficients. This coefficient relates the flow through the valve to the pressure drop that will be experienced. The coefficient is usually determined experimentally and reported on the manufacturer's literature for the valve. Control valves often will have coefficients that vary linearly or nonlinearly with flow. This is due to their variable openings. As a valve opens and closes the available flow area and amount of resistance changes. This is often not a concern for block valves, because they are only fully open or fully closed during normal operation, but this is not the case with control valves. The variation of the valve coefficient should be included in the model for control valves.

Other components in the system such as heat exchangers, scrubbers, filters, pipe reducers or elbows will also create pressure drop across the pipeline. The pressure differential of these elements is represented by segments of restrictive pipeline. The pipeline segment is tuned with its sizes and roughness to allow the respective flow and generate the required pressure drop. Once all the pipe, valve, and restriction elements are entered into the model, the system can be run. A result of the simulation will be the pressure and flows at various locations. These pressures and flows should be compared to actual operational data to ensure that the model is representative of the real system.

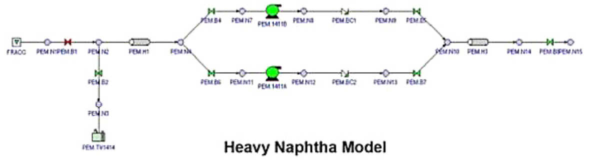
PIPELINE MODELING

Hydraulic Modeling

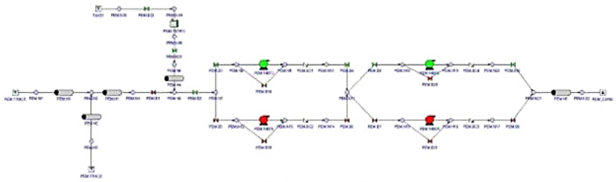
The quantitative analysis of each pumping station process system is based on fluid simulation using a pipeline or process simulation software. The process of creating a system model consists of a review of the available data for the system, developing a system model in the pipeline or process software, running the simulation based on given initial and boundary operating conditions, model validation and analysis of the resulting simulation data. These computational models allow for the evaluation and comparison of various conditions for a given system.

Computational Model

A computational model is developed in the pipeline or process software (Figure 3). Additional detailed data, specific to the equipment and piping for the process system being modeled, must then be input into the computational model. It is important that these data be accurate in order to ensure the predictive capability of the computational model. Defining the initial operating conditions as well as the operating constraints of the system are very important to limit the scope of the simulations and analysis. The model is then "tuned" such that the model accurately predicts known steady-state parameters for a given operating point. The model is validated by comparing the simulation results to known steady-state parameters at various other operating points.



Heavy Naphtha Model



Light Naphtha Model

Figure 3. Pump Station Computational Model.

Model Validation

The models for each pumping station are validated against steady-state data collected from the respective stations. Validation and verification of simulation models are a critical component of any physical simulation. Simulation software usually comprises algorithms that, over time, have been proven to predict the behavior of physical systems. These algorithms have a high numerical accuracy and are capable of making predictions on pipeline pressures and flows to within a few percent of the actual values. However, the validation of the pipeline models and input data plays a critical roll in the predictive accuracy of the simulation.

To validate the pipeline models of each station, steady-state flows and pressures are collected from each station to give a specified configuration. These will be compared to that same configuration replicated in the pipeline simulation. For the validation to be complete, several different operating points must be evaluated. Data should be collected for each unique operating condition. A baseline operating condition will be used to tune the pipeline model and account for losses that are not accounted for directly in the model. The simulation will be run at other operating points and the results will be compared to the collected data. This analysis will provide quantitative results as to the predictive capability of the pipeline models. In addition, model uncertainty can be calculated and used for the post-processing and analysis of the results.

Process System Simulation

Once a computational model has been developed and all pertinent data have been input into the model, simulations can be run at various operating points. The first few operating points are used to tune and later validate the computational model as described above. The simulation can provide both steady-state and transient predictions for a given configuration. Data from the process simulation is analyzed and possible design changes are proposed. These changes are then implemented into the model and evaluated. The evaluation of the modified model will determine if the recommended design changes are effective.

Computational Fluid Dynamic Modeling

CFD analysis is one of the branches of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the numerous calculations required to simulate the interaction of fluids and gases at certain conditions. The fundamental basis of any CFD problems is the Navier-Stokes equations, which define any single-phase fluid flow. These equations can be simplified by removing terms describing viscosity to yield the Euler equations. In all CFD simulations, the same basic procedure is followed. First the geometry of the problem is defined, then the physical modeling is defined and generated, and lastly, the boundary conditions are applied. After the model is fully defined, the simulation is started, and the equations are solved iteratively as a steady-state or transient problem. Finally, a postprocessor is used for the analysis and visualization of the solution. Many postprocessors can provide streamlines of flow, pressure gradients, temperature gradients, velocity gradients, and velocity vectors. CFD is used comprehensively for all turbomachinery and oil/gas piping technologies. Its use in pipes is usually focused on investigating blockages inside the pipe, effects of different shaped elbows, boundary layer and roughness effects of the pipe, etc.

The fluid analysis effort requires the development of a full, three-dimensional, computational fluid dynamics analysis of the pipeline system. Commercially available CFD codes are used to perform this type of analysis (Figure 4).

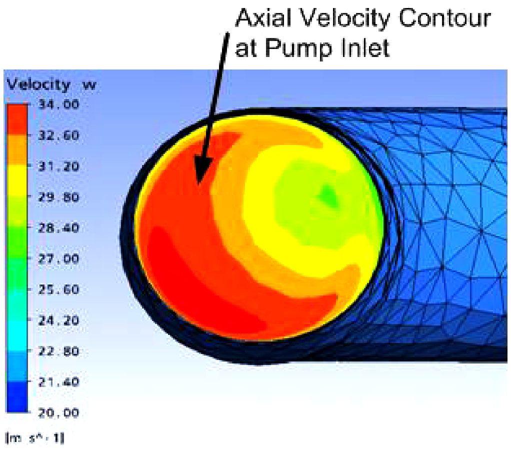


Figure 4. Example of CFD Flow Analysis Applied to a Pump Suction Piping.

CASES STUDIES

Hydraulic Modeling of a Liquid Petroleum Gas System

A process system analysis, optimization, and basic design services for liquids forwarding systems of different interconnected process plants were performed using a hydraulic simulation approach. The main objective of the analysis was to upgrade the pumping stations, improve existing operating philosophies and modify the hydraulic system to optimize and increase the capacity of the entire system. The interconnected pipeline system has four refining-storage-pumping facilities and one storage-pumping terminal facility. Within each of these process facilities, different liquefied petroleum gas (LPG) mixtures are being transported using a pressurized pumping and distribution system. All installations are connected via pressurized pipelines that vary in diameter from 12 inches up to 20 inches. The main 20 inch pipeline groups three pumping stations, which are separated by approximately seven to 81 miles. Then the main pipeline travels 172 miles in a relatively flat terrain up to a main booster station.

Two of the pumping stations are production facilities while the third station is principally the receiving/storage/terminal facility. The first pumping facility has six LPG booster pumps and six LPG main pumps. The pumps can operate in series (booster and main) or can operate with booster or main only. The second facility has five LPG booster pumps and four LPG main pumps. The LPG pumps can either feed a sales pipeline or a different pipeline from the main system.

Audits of the three facilities identified liquids forwarding operation, controls, instrumentation, performance, and safety issues. Other issues of concern are internal valve leaks, the need for better flow as well as pressure control, and transient effects between two of the nearby stations, which have resulted in pump shutdowns at one facility when a pump is started at the other station. Currently, at most locations, the control valves are manual. Therefore, a smooth operation for the station's transient events is difficult to achieve.

In addition, changes in operating conditions due to commercial requirements in the main pipeline system have been causing issues in the normal operation of the pumping facilities. The pumping-pipeline system was originally designed to operate at a fixed flow from each facility with almost no flexibility for flow fluctuations incorporated. Therefore, a redesign of the existing system was required to accommodate the new requirements. Figure 5 and Figure 6 show historical flow rate data for both pumping facilities. These data correspond to the last six months of operation and reflect the constant change in flow requirements.

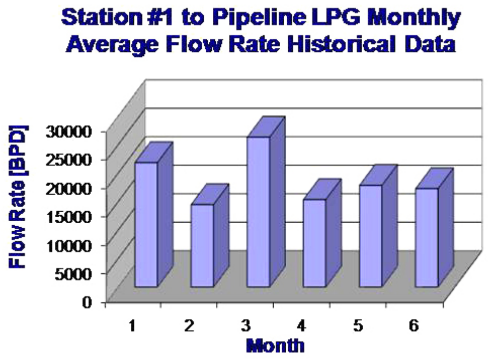


Figure 5. Average Flow Rate Data Station #1.

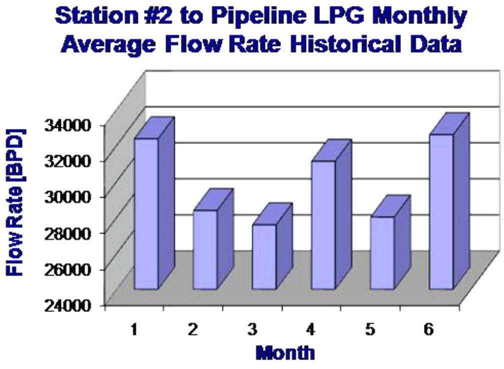


Figure 6. Average Flow Rate Data Station #2.

Pipeline Model Approach

The liquid forwarding process, redesign, and analysis were conducted using a pipeline modeling approach. In this analysis, all stations and their interconnected pipelines were modeled with commercial pipeline software. Existing pump data were included in the model for validation against real operating conditions. Then the models were used to propose different system improvements such as automatic control system, recirculation lines, better flow scheduling, and general pipeline operating philosophies. An example of one pumping station model is shown in Figure 7. The tuning and validation data were obtained through the existing SCADA system at different operating conditions. Potential redesign conditions were based on the new capacity requirements and they were compared with the historical data available.

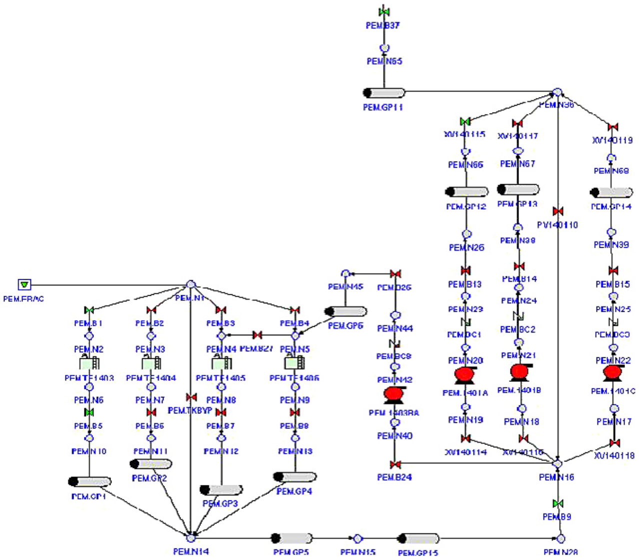


Figure 7. Example of a Pumping Station Model.

System curves were calculated for each facility considering their interaction with the pipeline system. In general, the flow and pressure supply by a station changed based on the operating conditions of the other station. Therefore, extreme operating cases were used as basis for the system curves of each system. Figure 8 presents the typical system curves and operating points for station #1. Different flow conditions were simulated for each station to define the operating philosophies of the entire system. The main criteria for determining the flow balance were storage capacity, pumping equipment availability, pipeline constraints (MAOP) and commercial requirements.

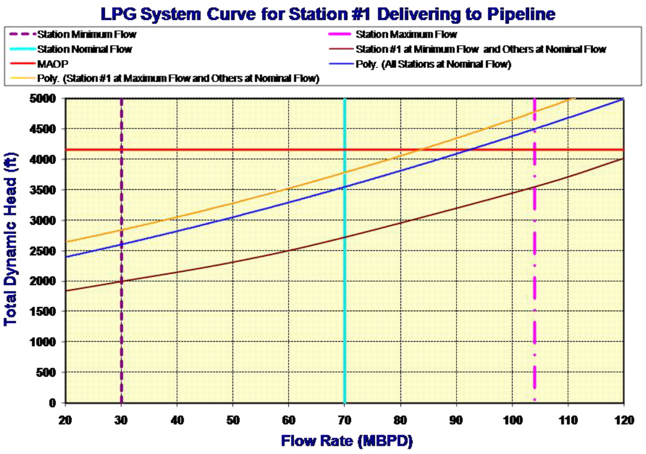


Figure 8. System Curves for Station #1.

It was found that the majority of the stations required an upgrade of their pumping equipment and control systems since they were controlling the system manually and had serious cavitation issues. The proposed changes included the use of variable frequency drives for almost all the primary pumps. This would provide a smooth start-up and shut down, and better flow control while saving energy and reducing the risk during flow transitions. Each selected new pump was modeled including the station system curves considering the expected operating ranges as shown in Figure 9.

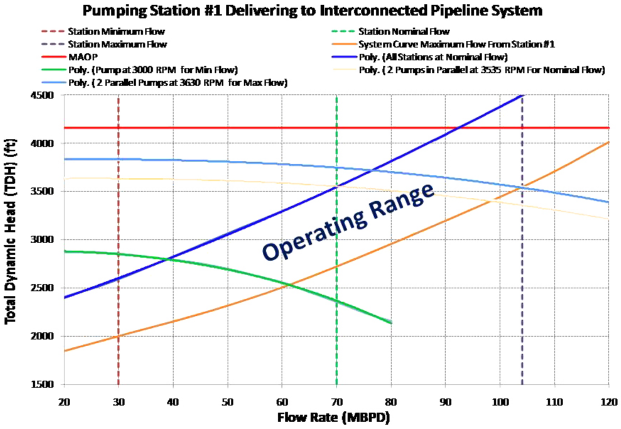


Figure 9. Pumping Station #1 System and Pumps Curves.

After obtaining all the system curves, defining the possible operating philosophies, and selecting the new pumps, the final simulation cases were run to verify the system performance and confirm its operability under some specific transient events.

As a summary in this analysis, the following activities were performed in chronological order: verification of the existing process data, confirmation of the capacity and capability of the equipment that was present, operating condition ranges and requirements were quantified, current operating conditions and any mismatches were noted, system areas and equipment that require improvements were identified, type of pumps that were required or beneficial were determined, and selection and validation of new pumping equipment were accomplished (Figure 10).

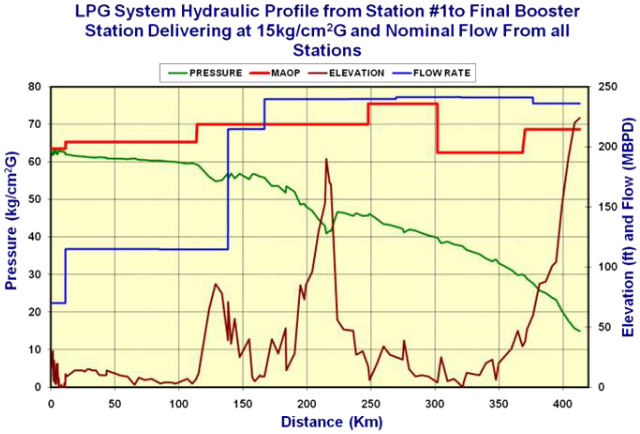


Figure 10. LPG System Hydraulic Profile.

CFD and Hydraulic Modeling for NPSH Determination in a Tank-Pump System

This case study focuses on an analysis that was conducted on a new pump installation. The pump was being installed to transfer water from a tank. There was concern that the NPSH would be below the required minimum in the desired configuration. The analysis was conducted with a combination of CFD and use of a pipeline simulator.

Analysis Approach

The objective of this analysis was to determine if the NPSHR would be met for the selected configuration of the piping system from the tank to the pump. The NPSHR was determined from the pump curves provided by the manufacturer, which was 30 ft of head at 8000 gpm. The configuration from the tank to the pump included a suction nozzle, several runs of piping, a reducer, and a valve (Figure 11). The pressure drop across the piping runs, valve, and reducer are to be estimated with a pipeline simulator. The pressure drop across the suction nozzle was estimated using CFD.

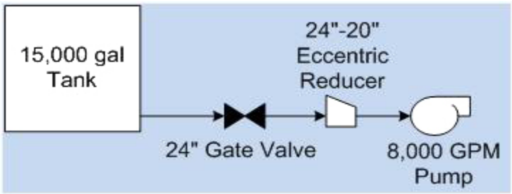


Figure 11. Schematic of System to be Analyzed.

Fluid Properties

Water is a well-known substance and the typical water properties were used for the analysis. The system was modeled at 85°F, which was an average temperature for the water used in the application.

CFD

The pressure drop across the suction nozzle was not known. This nozzle was designed and fabricated by the manufactures of the tank, but they did not provide the pressure loss. CFD was used to determine this value at the flow operating condition of 8000 gpm with a total of 12 feet of water above the centerline of the suction nozzle.

Models of the piping/pumping system can be generated in two ways for CFD. A model of the solid geometry can be constructed. This can then be imported into a CFD software. The CFD software will generate the mesh of the fluid inside of the solid geometry. Another method is to generate a solid model of the fluid. This allows the CFD to directly mesh the model imported. A solid model of the suction nozzle was generated in the tank using the later method. This solid piece was subtracted from the tank fluid in order to generate the fluid model for CFD (refer to Figure 12). The fluid level of the tank was set to the lowest level that could be achieved before the pump was set to shut-off. This will provide a worst case scenario result.

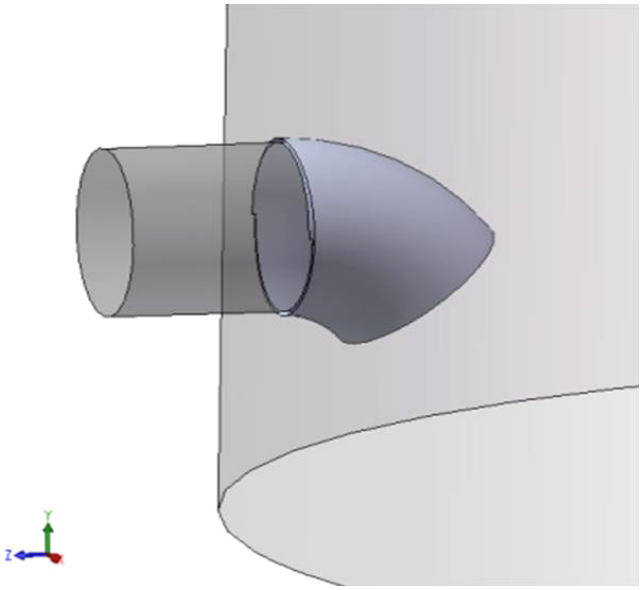


Figure 12. Fluid Model of Suction Nozzle in Tank.

Once the model is imported into the CFD solver, the boundary conditions are applied. The water temperature was set to 85°F, pressure on the top of the water to ambient pressure (14.69 psia), flow through the nozzle at 8000 gpm, and gravity was applied. Once the CFD model converged, the pressure gradient across the exit of the suction nozzle was plotted (Figure 13). This showed the pressure at the centerline to be 19.38 psia at the exit of the suction nozzle. The pressure at the centerline at the entrance to the nozzle was 19.89 psia. This equates to a pressure drop of 0.51 psia across the suction nozzle.

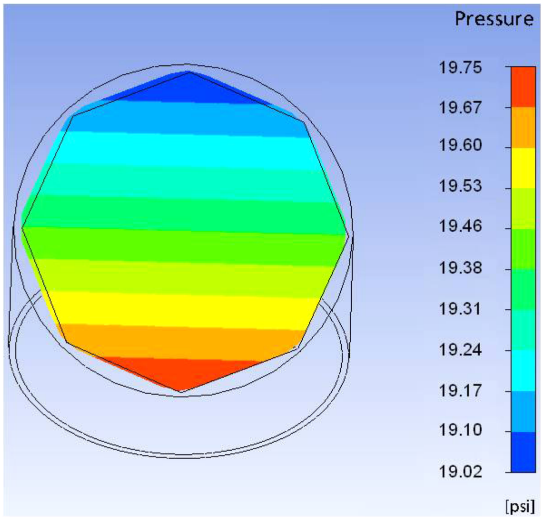


Figure 13. Pressure Gradient in Exit of Suction Nozzle.

Pipeline Simulator

The pressure drop across the remaining compoments of the piping system can be determined two ways: basic pressure drop hand calculations or with the use of a pipeline simulator. Due to the simplicity of this case study, hand calculations would be more efficient, but in more complex systems, a pipeline simulator would be beneficial. Therefore, the simulator was used to calculate the total pressure drop across the suction piping from the tank to the pump.

A model was generated containing piping, valve, tank, and pressure drop elements. A pump could be included in this model. However, since we were only modeling one condition, the pump was not included. The boundary conditions are set at each end. The tank has a flow of 8000 gpm entering it. The other boundary condition is at the exit of the last pipe (or where the entrance to the pump is), which was also set to 8000 gpm. The pressure at this point can be monitored during the simulation.

The piping was input with the inner diameter, length, and estimated surface roughness defined. The pressure drop elements are actually small pipe elements, but their surface roughness is tuned such that the pressure drop across that pipe element is equal to the pressure drop experienced. The first pressure drop element is the one obtained from the CFD for the suction nozzle in the tank. The valve is defined by its valve coefficient also known as the CV. The CV is calculated based on the size and type of valve. The CV for the 24 inch gate valve was 55,585 gal/min-psi$^{0.5}$. The equations for calculating this can be found in the Crane (2006) (Figure 14).

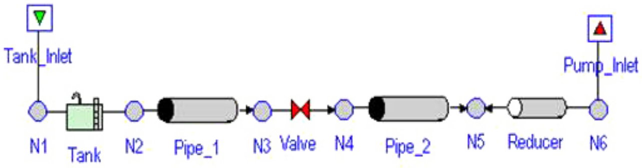


Figure 14. Model of System for Hydraulic Analysis.

Results

The pipeline simulation analysis showed a final pressure drop of 2.43 psia from the tank to the pump suction. The starting pressure at the centerline of the entrance of the suction nozzle was 19.89 psia. The pressure at the suction of the pump will be 17.46 psia. This equates to 40.29 ft of head. The NPSHR at the pump suction was 30 ft. The configuration of the system analyzed will meet the NPSHR requirements with a 10 ft safety margin.

CONCLUSIONS

During the design process, when troubleshooting, or retrofitting a system, modeling tools can be useful. The tools provide a method for simulating the system and determining how it will react to different operational conditions. Some examples of tools that are commonly used for this analysis are hydraulic modeling, CFD, and process simulators. Operational conditions that can be modeled include steady-state conditions, startups, shutdowns, process upsets, varying flows and pressure, pulsations, and other conditions. This paper reviewed how each of these tools are used and what are important parameters that must be considered when modeling these systems. Lastly, case studies were presented that provided examples of how an analysis would be completed.

It has been demonstrated that hydraulic modeling provides an excellent approach for engineering analysis of new or existing installations. Recommendations about how to approach different scenarios have been presented in this tutorial through a generic methodology and examples.

REFERENCES

Crane Technical Paper 410, 2006.

International Standard ISO 13709, 2007, “Centrifugal Pumps for Petroleum, Petrochemical, and Natural Gas Industries,” International Organization for Standardization, Geneva, Switzerland.

BIBLIOGRAPHY

Garcia, A., Brun, K., Aparicio, A., 2009, “Case Study of Liquids Drop-Out in a Natural Gas Pipeline Network,” Pipeline Simulation Interest Group, Galveston, Texas.

McKee, R. J. and Broerman, E., 2009, “Acoustics in Pumping Systems,” *Proceedings of the Twenty-Fifth International Pump Users Symposium*, Turbomachinery Laboratory, Texas A&M University, College Station, Texas, pp. 69-74.

Mohitpour, M., Golshan, H., and Alan, M., 2005, “Pipeline Design & Construction,” ASME Press, New York.

Mohitpour, M., Szabo, J., and Hardeveld, T., 2005, “Pipeline Operation & Maintenance—A Practical Approach,” ASME Press, New York.

Santos, S. P., 2008, “Availability and Risk Analysis Effects on Gas Pipeline Tariff Making,” Seventh International Pipeline Conference, Calgary, Alberta, Canada.

Santos, S. P., 2009, “Monte Carlo Simulation—A Key for a Feasible Pipeline Design,” Pipeline Simulation Interest Group, Galveston, Texas.

Southwest Research Institute Applied Physics Division Staff, 1982, “Controlling the Effects of Pulsations and Fluid Transients in Industrial Plants,” SGA-PCRC Seminar, San Antonio, Texas.