Proceeding of ETCE/OMAE2000 Joint Conference
Energy for the New Millenium
February 14-17, 2000, New Orleans, LA

PR-025

NUMERICAL SIMULATION OF THE FLOW THROUGH AN INTERMITTENT GAS LIFT VALVE

Maria A. Concho and Sandro Gasbarri
conchom@pdvsa.com and gasbarris@pdvsa.com
PDVSA-INTEVEP, Venezuela

Luis R. Rojas-Solórzano
rrojas@usb.ve
Universidad Simón Bolívar, Venezuela

ABSTRACT

The numerical simulation of the flow through a pilot valve used in the oil intermittent gas lift process is presented. The complexity of the non-isothermal compressible flow is modeled by the solution of the Navier-Stokes, Mass Conservation and Energy equations for the compressible flow. Numerical results and analyses pertaining to the flow dynamics through a 1½-inch pilot valve at an operating condition encountered in typical field operations are presented.

INTRODUCTION

Since 1960 many researchers have studied the Intermittent Gas Lift process. In the last decade there have been important advances related to the development of a mechanistic model [1] and field scale tests [2]. However, pilot valve behavior has not been incorporated to the existing models, leading to inaccuracies in gas flow calculations.

For many years it has been a common practice to use the Thornhill and Craver equation, originally developed for flow beams, to compute for pilot valve gas flow. However it can be shown that there is considerable difference between calculated values and actual field data. This situation brought up the need for correlating gas flow more accurately in order to improve intermittent gas lift simulations. This is the subject of the present study, it will present a new approach to gas flow calculations for the pilot valve.

It was not until 1999 when intermittent pilot valve studies were conducted: one at Tulsa University [3] and the one, which is subject of this paper.

Over the last years there have been many studies for developing accurate correlations to predict flow through continuous gas lift valves, mainly from Tulsa University [4-9]. These studies were made in a test facility especially designed for this purpose. Later, Milano [3] used that facility to test a 1-inch pilot valve. He developed a correlation to predict flow through the valve as a function of upstream and downstream pressure by correcting the well-known Thornhill and Craver equation[10]. Milano also verified closure pressure conditions and developed a correlation to predict closing pressure.
However, a 1½-inch pilot valve correlation has not been developed yet. Moreover, there is a lack of physical understanding of the compressible flow behavior within the pilot valve. Thus, a preliminary study is presented here using numerical simulation to analyze flow details that are not currently available from experiments. This study will provide valuable insight to understand current differences between gas flow calculations and actual field data.

MODEL DEVELOPMENT

Pilot Valve Configuration

A Pilot valve is a gas lift valve especially designed for intermittent applications. Its configuration consists of two sections: a pilot section and a power section. The power section is connected to the pilot section, so it opens and closes after the pilot section opens and closes, respectively. Figure 1 shows a typical pilot valve configuration.

The pilot section most important elements are the bellows, pilot ball and spring. When the pilot section is closed, the ball seals and no gas flow is allowed through. The spring is calibrated in order to give the appropriate opening and closing conditions. The opening condition is established when the forces exerted by the casing pressure and tubing pressure overcome the resistance of the spring. The tubing pressure acts in the opening by means of a bleed hole located in the power piston which allows fluid to fill up the space below the seat.

Once the pilot section opens, gas is allowed through the seat, the power piston moves downward until it opens the power section to gas flow, and finally gas flows into the tubing. During a normal intermittent operation, as the gas flow is established, the casing pressure gradually decreases, leading to the closure of the pilot and power section.

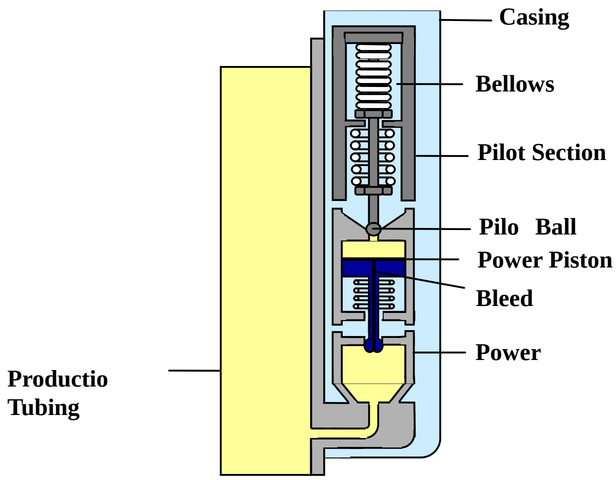

Fig.1. Pilot Valve configuration

PHYSICAL MODELING

The domain considered in this study is shown in Fig. 2. Minor simplifications made on actual geometry were based on the following assumptions:

- There is no gas flow through the bleed hole. In fact, the flow established through the bleed hole is not of great significance compared to the total flow. See Point 1 in Fig. 2.
- The upper boundary is located on the wall that separates the power piston and the power section gas entrance. Thus, the little amount of gas that flows through a small hole right on the boundary wall and fills up a space within the piston is neglected. This negligible amount of gas remains mainly stationary and does not contribute significantly to the main flow. See Point 2 in Fig. 2.

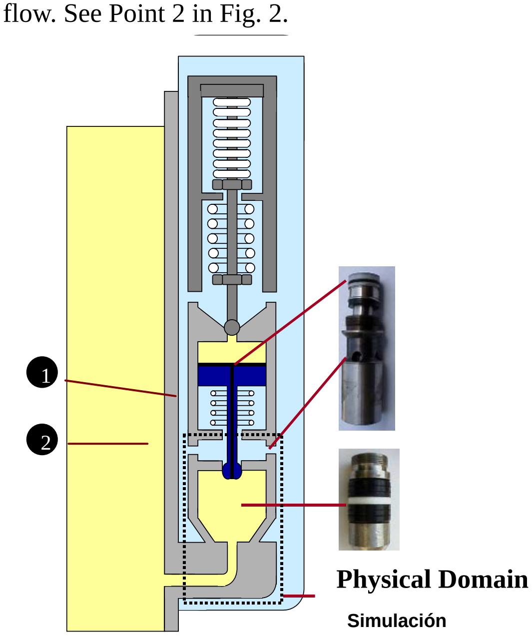

Fig. 2. Physical domain for gas flow study

The computational domain used for actual simulations with CFX™ solver is depicted in Fig. 3. It corresponds to ¼ of the total valve geometry due to flow symmetry.

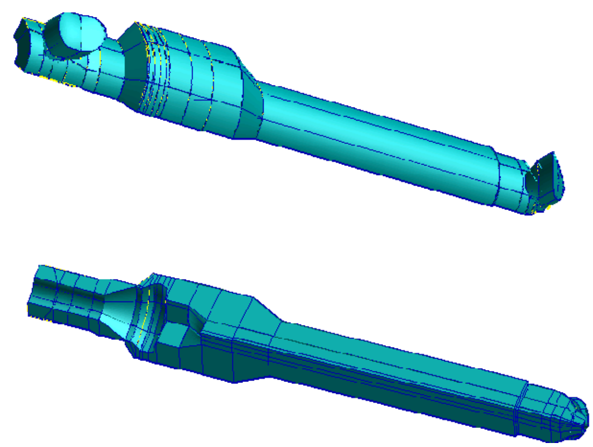

Fig. 3. Computational domain

.

MATHEMATICAL MODEL

Model Assumptions

Mathematical equations to be solved for, were derived from momentum, energy and mass conservation along with the following considerations:

- Newtonian fluid
- Turbulence is modeled through the k-ε model as developed by Launder and Spalding [11].
- One-phase flow (natural gas). This assumption involves that the power section is opened and all fluid initially inside the valve has already been displaced by the gas flow.
- The constitutive relation for $\rho = \rho\left(T, p\right)$ is modeled using ideal gas equation of state.
- A constitutive equation is used to relate enthalpy with temperature and pressure.
- There is no heat flux through wall boundaries.

Boundary Conditions

Boundary conditions are in the form of:

- Upstream Pressure (Casing Pressure)
- Downstream Pressure (Production Tubing Pressure)
- Upstream Temperature

Solution: Finite Volume Method

The mathematical equations, derived from the proposed model, are solved for using a state-of-the-art code based on the finite volume method[12].

The finite volume method subdivides the problem domain into control volumes. The governing differential equations are then integrated for each control volume leading to a discrete equation which relates the variable in the center of the control volume to its surrounding nodes.

The governing differential equations are essentially mass, momentum and energy conservation. The energy equation is expressed in terms of enthalpy. Momentum and energy equations can be expressed as:

Transient+Convection-Difussion=Source

The solver discretizes all but the convective term, through second order central differences. For the convective term the UPWIND scheme is used while the transient term is discretized using backward differences.

The non-lineal nature of the governing equations involves an iterative procedure. The SIMPLE algorithm is used to couple the pressure and velocity terms in the momentum and mass conservation equations. More details can be found in reference [13].

The compressible flow model within the code has been extensively validated. Results obtained for compressible flow through a convergent divergent nozzle, not shown here, demonstrated good agreement with theory.

RESULTS AND ANALYSIS

Results proved to agree reasonably well with test data. Average values obtained from a test performed on a field-scale well[2] were compared with simulated values. Results so far showed around 14% differences on mass flow rate values, which are expected to occur due to current assumptions. This is a considerable improvement over the Thronhill and Craver equation, where equivalent calculations have shown up to 200% error in mass flow rate (10% error in discharge pressure).

As part of future work, fluid properties will be further calibrated against test values in order to develop a correlation for gas flow through the valve. Real gas equation of state will be included by means of an external subroutine.

The results were validated against mesh refinement leading to differences under 1%. The following results correspond to an operating condition with the valve slightly opened and casing pressure and tubing pressure of 486 and 436 psi respectively.

Velocity Field

Velocity (module) field is plotted in Fig. 4. The highest velocity is encountered in the throat. The throat corresponds to the gap around the power piston where fluid accelerates.

Details of flow speed at the entrance and exit sections are depicted in Figs. 5 and 6.

Mach Number

Figure 7 shows Mach number contours near the throat region. For the analyzed operating condition, Mach numbers correspond to incompressible flow with the highest Mach number located in the throat.

Temperature Field

Figure 8 shows temperature field details within the throat region. Temperature values are in degrees Kelvin.

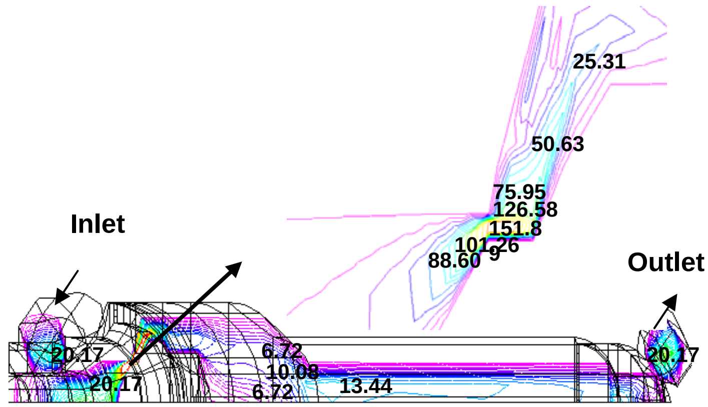

Fig. 4. Speed [m/s] contours at longitudinal plane (geometry not to scale)

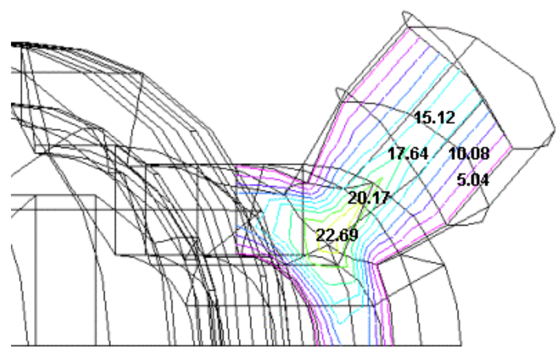

Fig. 5. Speed [m/s] contours at entrance longitudinal plane

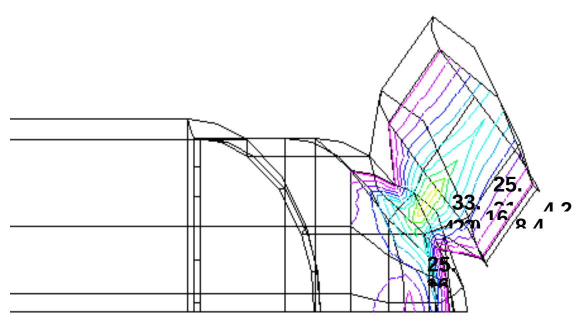

Fig. 6. Speed [m/s] contours at exit longitudinal plane

Results agree with physics of the problem, explained through thermodynamics first law.

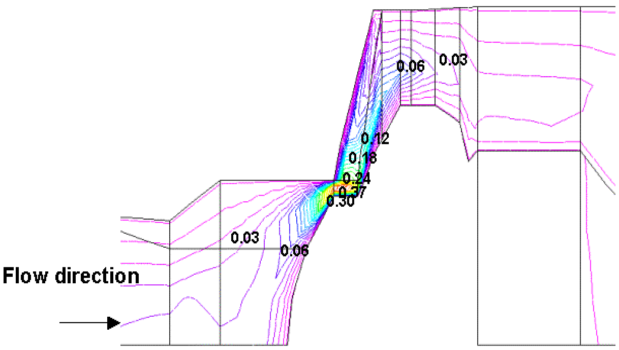

Fig. 7. Mach number contours around the throat region

As no heat and work is added to the system, and considering steady state, first law reduces to:

$$\dot{m}\left(u+\frac{p}{\rho}+\frac{V^2}{2}+gz\right)_i-\dot{m}\left(u+\frac{p}{\rho}+\frac{V^2}{2}+gz\right)_o=0$$

where:
m: mass flow rate
p: pressure
ρ : density
$\frac{V^2}{2}$: Kinetic energy
u: internal energy
gz: potential energy

By relating density to pressure and temperature using perfect gas law, internal energy to temperature and neglecting potential energy changes, it can be stated that temperature would change to overcome velocity changes. This explains how there is a cooling effect equivalent to a 2% temperature reduction in the throat where fluid accelerates. And later, where fluid decelerates, it recovers temperature.

Pressure Field

Figure 9 shows the pressure field for a longitudinal plane and details of the throat region. Pressure values are in Pa.

As fluid flow regime is subsonic, it approximates incompressible flow behavior where pressure can be inversely related to velocity. As fluid accelerates in the throat region, the pressure decreases. This explains how the minimum value of pressure corresponds to the maximum velocity. This minimum pressure corresponds to 13% of inlet pressure.

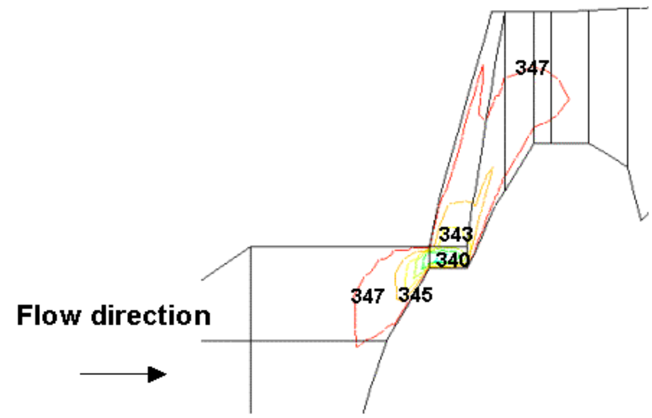

Fig. 8. Temperature [°K] contours around the throat region

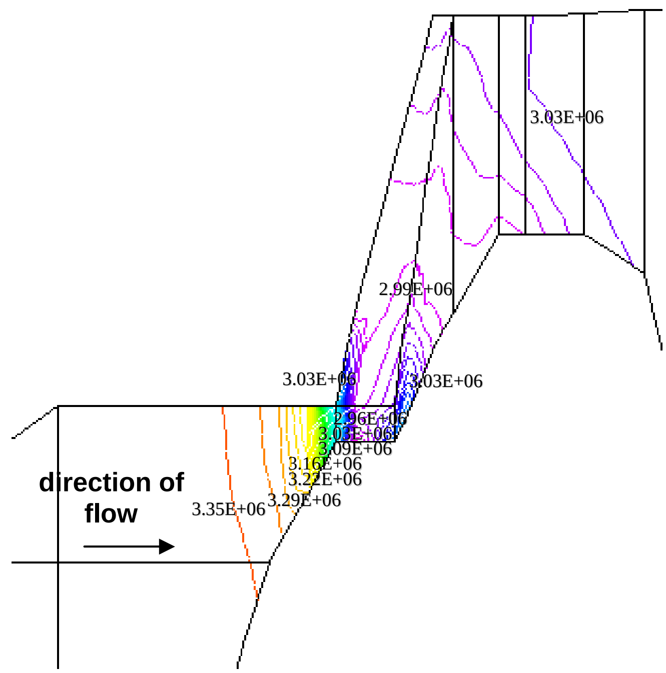

Fig. 9. Pressure [Pa] contours around the throat region

CONCLUDING REMARKS

The preliminary numerical simulation of the compressible flow through a gas lift valve at typical operating open condition is performed.

Results permit to state that it is feasible to simulate fluid dynamics and heat transfer for the complex geometry encountered in a gas lift pilot valve.

The important role of valve throat design is demonstrated since this region develops the largest velocity, temperature and pressure gradients. Thus, differences between state-of-the-art gas flow calculation for pilot gas lift valves and actual field values may be explained from the physics related to the flow around the power piston where the highest velocity, pressure and temperature gradients are developed.

Current simulations are oriented towards the study of larger flow rates, typically found during opening stages. These conditions are quite challenging since it is expected to find shock-waves as the flow might become supersonic within the throat.

ACKNOWLEDGMENTS

The authors wish to thank PDVSA-Intevep for supporting this study. They would also like to thank the many colleagues in PDVSA-Intevep Gas Lift Department who provided valuable inputs and suggestions for this study.

REFERENCES

1. Liao, T., Schmidt, Z., Doty, D.: "Investigation of Intermittent Gas Lift by Using Mechanistic Modeling", paper SPE 29454 (1995).
2. Hernández, A., Gasbarri, S., Machado, M., Marcano, L., Manzanilla, R., y Guevara J.: "Field Scale research on Intermittent Gas Lift", SPE 52124 (1999).
3. Milano, P.: "Dynamic Performance of The Intermittent Gas-Lift Pilot Valve", M.S. Thesis, The University of Tulsa, OK (1999).
4. Biglarbigi, K.: "Gas Passage Performance of Gas-Lift Valves", M.S. Thesis, The University of Tulsa, OK (1985).
5. Hepguler, G.: "Dynamic Model of Gas-Lift Valve Performance", M.S. Thesis, The University of Tulsa, OK (1988).
6. Nieberding,M.: "Normalization of Nitrogen Loaded Gas Lift Valve Performance Data", M.S. Thesis, The University of Tulsa, OK (1988).
7. Sagar, R.K.: "Improved Dynamic Model of Gas Lift Valve Performance", M.S. Thesis, The University of Tulsa, OK (1991).
8. Bertovic, D.: "A Unified Model for Gas-Lift Valve Performance Incorporating Temperature Effects", M.S. Thesis, The University of Tulsa, OK (1995).
9. Faustinelli, J.: "Temperature and Flow Performance Modeling of Gas-Lift Valves", M.S. Thesis, The University of Tulsa, OK (1997).
10. Cook, Jr., Dotterweich, F.: "Report on the Calibration of Positive Flow Beans as Manufactured by GH –Thornhill Craver Company", Texas College of Arts and Industries, Kingsville, Texas (1946).
11. Launder, B.E., Spalding, D.B.: "The Numerical Computation of Turbulent Flows," *Computational Methods in Applied Mechanics and Engineering*, vo. 3, pp. 269-289. (1977).
12. CFX-4.2, "Solver Manual", AEA Technology Engineering software,Inc. (1997).
13. Patankar, S., " Numerical Heat Transfer and Fluid Flow", New York (1980).