Proceedings of ICFDP7:
Seventh International Congress on Fluid Dynamics and Propulsion
December 18-20, 2001, Cairo, Egypt

ICFDP7-2001027

MULTIPHASE CFD OF GASOLINE FLOW IN A NOVEL FUEL PIPE SYSTEM

Bhavin V. Mehta, Ph.D.
Associate Professor
Department of Mechanical Engineering
Ohio University, Athens, Ohio 45701
E-mail: mehta@ohio.edu, Phone: (740) 593-2739

ABSTRACT

The main aim of the project was to analyze the multiphase flow of gasoline, air and vapor during pumping of fuel in fuel pipes at the gas stations, and come up with the optimum design/shape for minimum turbulence which in-turn would minimize vapor generation. The analysis was a multiphase analysis with benzene, air and vapor as the phases. To this day, fuel tank pipes (pipe leading to the gas tank) have been made of aluminum or steel. The new design, which is being introduced to reduce the vapor produced by gasoline, consists of two polymer pipes, one inside the other, with a gap between the two pipes. Aluminum or steel pipes would be hard to bend inside an existing bent pipe. Since the inside pipe is bent into an existing pipe, convolutes are provided on the inside pipe to reduce stresses and provide flexibility. The new design will allow the vapor produced inside the main fuel pipe, and the tank, to be sent back (sucked back) through the area between the two pipes. A valve is provides at the entry section of the pipes and the vapor will be sucked in at the inlet by venturie effect, when the gasoline is pumped to the tank. This allows the vapor produced inside the main fuel pipe to be sent back at the inlet through the area between the two pipes and does not escape to the environment. An Automotive Company (Funding Source) provided two pipe/hose geometries to be considered for flow analysis and inner pipe design (Pipe A and Pipe B).

INTRODUCTION

Gasoline is made up of different hydrocarbon compounds, including aromatics, olefins, and benzene, all of which contribute to ozone and toxic air pollution. Ground-level ozone causes health problems because it damages lung tissue, reduces lung function, and sensitizes the lungs to other irritants. Scientific evidence indicates the ambient levels of ozone not only affect people with impaired respiratory systems, such as asthmatics, but healthy adults and children as well.

To improve engine performance, the volatility of gasoline, which is defined as the ability of gasoline to vaporize and mix with air under a variety of engine operating and ambient temperature conditions, is an important factor. However, gasoline with a high volatile content will result in greater emissions to the atmosphere

The 1990 Clean Air Act Amendments (Act) signed into law on November 15, 1990 deal with air pollution of a hazardous or toxic nature. The program aims at substantially reducing emissions from stationary sources, motor vehicle-related air pollution. In order to carry out the mandates of the Act of 1990 concerning the assessment and control of toxic pollution, the Environmental Protection Agency (EPA) must organize their technical and administrative resources and develop appropriate regulations. For the mobile source, EPA specifies the means of reducing carbon monoxide and hydrocarbon emissions from motor vehicles and refueling. Tailpipe standards for normal operation of cars have been established for non-methane hydrocarbons, CO and NOx. The Act regulations were subsequently negotiated with industry, federal and state governments, and environmental and consumer groups, which resulted in cleaner burning gasoline.

The cleaner gasoline is called reformulated gasoline (RFG). This has lower levels of certain compounds that contribute to air pollution. RFG will contain “chemical oxygen” (oxygenates) for octane enhancement. RFG will have lower volatility during the summer months. This means that it does not evaporate as easily as conventional gasoline. Even though this reduces the amount of gasoline vapors being produced it is not the final solution. The automobile industry is doing research to come up with a solution where the gas vapors are not exposed to the atmosphere and humans while filling up the gas tank of a car. The vaporizing of gasoline has led to the research in reducing vapor generated when an automobile is filled. This research is conducted by several companies including Ford, Chrysler and General Motors and other manufacturing companies associated with them.

INTRODUCTION TO MULTIPHASE MODELS

FLUENT is a general-purpose computer program for modeling fluid flow, heat transfer, and chemical reaction. Using FLUENT you can quickly analyze complex flow problems even if you do not have prior expertise in computational fluid dynamics or computer programming. FLUENT enables you to apply computer simulation methods to analyze and solve your practical design problems. FLUENT incorporates up-to-date modeling techniques and a wide range of physical models for simulating numerous types of fluid flow problems. These are accessible to you through an interactive graphical user interface for problem definition, computation, and graphical post-processing. When required, FLUENT can also be customized to your specific modeling needs and/or interfaced to your in-house CAD system.

A large number of flows encountered in nature and technology are a mixture of phases. Physical phases of matter are gas, liquid, and solid, but the concept of phase in a multiphase flow system is applied in a broader sense. In multiphase flow, a phase can be defined as an identifiable class of material that has a particular inertial response to and interaction with the flow and the potential filed in which it is immersed. For example, different-sized solid particles of the same material can be treated as different phases because each collection of particles with the same size will have a similar dynamical response to the flow field.

In single-phase flow, the models of laminar and turbulent flows are different. Laminar flows can be described by instantaneous quantities, the solutions of Navier-Stokes equations, while turbulent flows are explained by time or statistical averaged quantities which are the solutions of a system involving the Reynolds equations and some closure equations. Likewise, two-phase flow patterns must be known for modeling the physical phenomenon as closely as possible. It is impossible to explain bubbly flow and annular flows with the same model correctly.

Advances in computational fluid mechanics have provided the basis for further insight into the dynamics of multiphase flows. Currently there are two approaches for the numerical calculation of multiphase flows: the Euler-Lagrange approach and the Euler-Euler approach.

The Lagrangian Dispersed Phase Model

The Lagrangian dispersed phase model in FLUENT follows the Euler-Lagrange approach. The fluid phase is treated as a continuum by solving the time-averaged Navier-Stokes equations, while the dispersed phase is solved by tracking a large number of particles, bubbles, or droplets through the calculated flow field. The dispersed phase can exchange momentum, mass, and energy with the fluid phase. A fundamental assumption made in this model is that the dispersed second phase occupies a low volume fraction, even though high mass loading is acceptable.

The Euler-Euler Approach

In the Euler-Euler approach, the different phases are treated mathematically as interpreting continua. Since the volume of a phase cannot be occupied by the other phases, the concept of phasic volume fraction is introduced. These volumes fractions are assumed to be continuous functions of space and time and their sum is equal to one. Conservation equations for each phase are derived to obtain a set of equations which have similar structure for all phases. These equations are closed by providing constitutive relations which are obtained from empirical information, or, in the case of granular flows, by application of kinetic theory. In FLUENT, two different Euler-Euler multiphase models are available: the VOF (Volume of Fluid) Model and the Eulerian Multiphase Model.

The VOF Model

In the VOF model a single set of transport equations is solved, and the phases do not mix. The interface between phases is defined by solution of a transport equation and diffusion across the interface is prevented. Applications of the VOF model include stratified flows, free-surface flows, filling, sloshing, and jet breakup (surface tension).

The Eulerian multiphase model

The Eulerian multiphase model solves a set of *n* momentum, enthalpy, continuity, and *m* species for each phase. Coupling is achieved through the pressure and interphase exchange coefficients. The manner in which this coupling is handled depends upon the type of phases involved; granular (fluid-solid) flows are handled differently than non-granular (fluid-fluid) flows. For granular flows, the properties are obtained from application of kinetic theory. Momentum exchange between the phases is also dependent upon the type of mixture being modeled. FLUENT's user-defined subroutine capability allows customization of the calculation of the momentum exchange and heat exchange.

ANALYSIS RESULTS OF FLOW IN PIPES WITH CONVOLUTES

In the first part of the research, convolutes were modeled using the roughness parameter available in Fluent Software (Fig. 1), and compared to a model of pipe with convolutes (Fig. 2). Comparing the results, one could observe that after introducing a rough wall to substitute for convolutes, the turbulence decreased from $31.11/s^3$ to $15.37\ m/s^3$. This shows a high error of 35%, which meant that the roughness factor parameter was not adequate to represent convolutes and that convolutes will have to be created in the modeling stage and taken into account during the analysis. From the Table one can observe that for both straight and bent pipes the turbulence was increasing with the increase in convolutes. Extra care was taken so that enough convolutes would be provided to obtain the required flexibility in the pipe. A secondary phase of air was introduced in the model. The model was analyzed with different quantities of air; 20%, 25% and 30%. A similar analysis was done with Pipe B which had more convolutes than Pipe A. The results show that the turbulence decreased with an increase in air quantity. From the table one can also observe that the turbulence is actually decreasing with the increase in the percentage of air in pipe A and varies between 31.11 and $21.31\ m/s^3$. Similar behavior was observed in pipe B. Since the location of the outer pipe is predefined, only small changes could be made in the bends of the inner pipe. The pipe was analyzed with different bends and a different arrangement of convolutes (Fig. 2), and the pipe geometry with the minimum turbulence was selected for multiphase flow analysis. The detailed results of these studies will be presented.

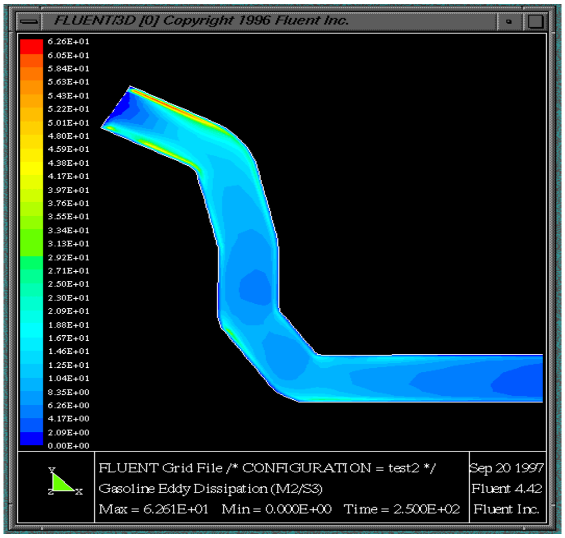

Fig 1: Pipe without Convolutes (Air 0.1 and Vapor 0.1)

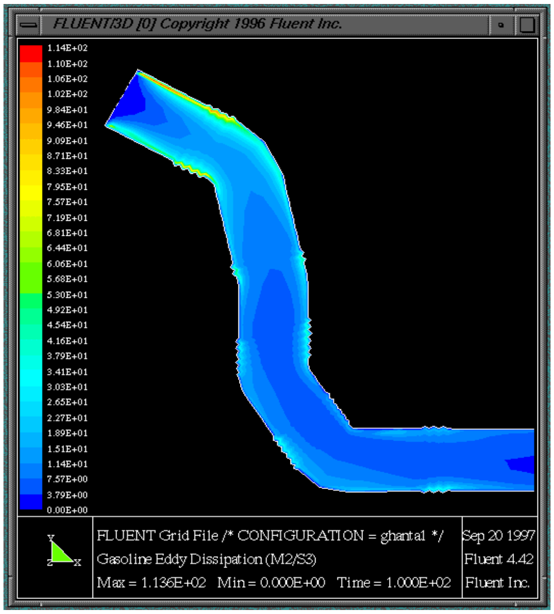

Fig 2: Pipe with Convolutes (Air 0.1 Vapor 0.1)

Table 1: Summary of Results

	CASE	MAXIMUM EDDY/ TURBULENCE (m^2/s^3)	
1)	**Original Pipe**	***Pipe A***	***Pipe B***
		31.11	38.94
2)	**Pipe with Roughness**	15.37	
3)	**Multiphase Models**	Pipe A	Pipe B
	20% Air & 80% Gasoline	26.19	37.90
	25% Air & 75% Gasoline	24.09	34.83
	30% Air & 70% Gasoline	21.31	31.06
4)	**Pipe with bends**		
	Pipe 1	32.76	
	Pipe 2	40.2	
	Multiphase Models	Pipe A	Pipe B
	20% Air & 80% Gasoline	25.64	24.8
	25% Air & 75% Gasoline	22.87	21.7
	20% Air, 70% Gasoline & 10% G.Vapor	20.35	
	25% Air, 65% Gasoline & 10% G.Vapor	18.88	

CONCLUSION

The model was estimated to be in 2-D as there was very little deviation of the geometry in the third dimension. This helped in the overall analysis process as the pipe was symmetrical and could easily be modeled with relative ease.

After analyzing the bent pipe 1 with different quantities of air at the inlet, one can observe that the turbulence decreases with increase in air (Table 1). For example on adding a 0.1 volume fraction of gasoline vapor the turbulence decreased from 25.64 to 20.35 m/s^3. A more accurate solution would be provided by a 3-D analysis of the problem. This could be done in the near future on super computers.

This research is just an estimate and the whole fuel system should be studied with the inlet and outlet valves to get an accurate solution. More study should also be done in the area of multiphase flows in bent pipes with convolutes. FLUENT can only calculate the phase change with change in temperature but not with turbulence. A more accurate solution would be provided by a 3-D analysis of the problem. Thus could be done in the near future on Supercomputers. FLUENT's user-defined subroutine capability allows customization of the calculation of the momentum exchange and heat exchange. By adding these subroutines we could predict vapor phase relating it to turbulence and temperature change. A mathematical model has to be developed and incorporated in FLUENT to predict the phase change and hence calculate the amount of vapor generated.

REFERENCES

1. H. Imai et al., "Analysis of Transient Phenomena in Gas and Two Phase Pipelines", Nippon Steel Technical Report, (1994), No. 60. p25-31.

2. K. Minami et al., "Transient Two-Phase Flow Behavior in Pipelines-Experiment and Modeling", Int. J. Multiphase Flow, (1995), p739-752.

3. E. Foumeny et al., "Can CFD improve the handling of air, gas and gas-liquid mixture", Chemical Eng. Progress v. 89 (Feb. '93), p21-26.

4. K. Kohda et al., "Analysis of transient gas-liquid two-phase flow in pipelines", Journal of Energy Resources technology v. 110, (June '88), p93-10.

5. D. Wilcox, "Turbulence modeling for CFD", AIAA Journal v.32, (June'94), p1339.

6. GeoMesh, User's Manual, Fluent Inc. Lebanon, NH, (1996).

7. Fluent, User's Manual, Fluent Inc. Lebanon NH, (1996).

8. Vehicle Fuels and the 1990 Clean Air Act, EPA 400-F-92-015, (August '94).