THE ANNALS OF "DUNAREA DE JOS" UNIVERSITY OF GALATI
FASCICLE IX. METALLURGY AND MATERIALS SCIENCE
N°. 4 - 2023, ISSN 2668-4748; e-ISSN 2668-4756
Article DOI: https://doi.org/10.35219/mms.2023.4.07

FLOW SIMULATION OF FLUID UNDER PRESSURE, THROUGH PIPES FOR OIL AND GAS TRANSPORT

Beatrice TUDOR, Mirela NOUR
"Dunarea de Jos" University of Galati, Romania
e-mail: beatrice.bodor@ugal.ro

ABSTRACT

The work presents a simulation of the flow of liquids under pressure, through pipelines intended for the transport of oil and natural gas. The simulation was done using the SOLIDWORKS program. Computational Fluid Dynamics (CFD) simulation facilitates the analysis of complex fluid flow problems involving liquid-gas, fluid-solid, or fluid-fluid interactions. CFD allows us to design products and systems that meet fluid flow and heat transfer requirements.

KEYWORDS: pipes, simulation, fluid flow

1. Introduction

Romania has a gas exploitation industry developed on the basis of its own resources, ranking among the top 12 gas industries in the world. But due to a long period of exploitation (over 90 years), the natural flow and pressure decline is felt, as in any extractive industry.

The natural gas industry, both extraction and transport, presents a series of risks for the workers in these activities (work accidents), for the population of neighbouring towns (major accidents) and for the environment [1, 3].

The process of gas transportation through pipelines and their distribution to consumers is subject to specific regulations in all the member countries of the European Union.

The legislation in force aims to create distribution networks, dimensioned according to the source-consumption balance, which will ensure the prospects for the development of consumption and supply over time.

From the point of view of the technological elements within the production, transport, distribution and use of natural gas, pipelines fall into the following pressure ranges:

• high pressure > 6 bar (collector, transport pipes and technological installations related to oil sites);
• medium pressure, between 2 and 6 bar (gas supply systems);
• low pressure, between 0.05 and 2 bar (gas supply systems);
• low pressure < 0.05 bar (gas supply systems) [2].

2. Flow simulation of liquids under pressure through pipes

The flow simulation was carried out using the SOLIDWORKS program.

Computational Fluid Dynamics or CFD, is a technique that deals with the analysis and study of fluid flow fields through numerical analysis. SOLIDWORKS is a CFD software designed to provide dynamic feedback on fluid flow. It has parametric optimization capability, and can automate the design and analysis process to discover the best design in SOLIDWORKS CAD.

Computational Fluid Dynamics (CFD) is an analysis tool for multiphysics systems that simulates the behavior of fluids taking into account their thermodynamic properties, using numerical models [4].

CFD simulation facilitates the analysis of various complex fluid flow problems involving liquid-gas, fluid-solid or fluid-fluid interactions. The program uses advanced solutions to transform physical laws from differential equations into algebraic equations, and to solve them as efficiently as possible. Engineers and analysts use computers to simulate the free-flowing fluid and its interaction with surfaces.

CFD simulations are widely used in many fields to analyse fluid flow and heat transfer of design components, to inform design decisions and ultimately to bring products to market in a timely

N°. 4 - 2023, ISSN 2668-4748; e-ISSN 2668-4756
Article DOI: https://doi.org/10.35219/mms.2023.4.07

manner. shorter. CFD software provides various tools for optimizing system design by simulating fluid flow and its free movement over surfaces.

System-level design requires optimization of fluid behavior and material strength. SOLIDWORKS is a 3D CAD (Computer Aided Design) design software for modeling 3D, 2D plans and assemblies. The software offers a wide range of solutions for designing, simulating, manufacturing, publishing and managing design process data [6].

This program allows the simulation of different liquids, gases and oil, for different engineering scenarios. Some common applications involve flow through manifolds, heat exchangers, electronic cooling. Fluid-structure interactions (FSI) refers to the interaction between a fluid flow and a solid structure.

When a fluid flows around or through a solid object, it can exert forces on the object, which can cause it to deform or vibrate. This involves deformation or vibration of the object and can affect the flow of the fluid, leading to changes in the behavior of the fluid.

For certain applications, the fluid in a project may contain solid or liquid particles. The behavior of these particles, and the fluid that interacts with them, is essential to the operation and effectiveness of the system. With the help of the SOLIDWORKS program, we can analyse the behavior of the particles and we can see results, regarding the erosion and accumulation of the particles in the system. Based on the material properties, high flow and other physical characteristics of the system design, this can be analysed.

A combination of fluid dynamics and mechanics is used to analyse FSI. Computational fluid dynamics (CFD) can be used to model the fluid flow, while finite element analysis (FEA) can be used to model the solid structure. The coupling of these two simulations allows studying the effects of FSI [5].

FSI analysis is important for the design and optimization of structures and systems that interact with fluids. By modeling and analysing FSI, engineers can optimize the design of structures to reduce the effects of fluid-induced vibration or deformation, and improve the performance and safety of these structures.

Computational fluid dynamics is the simulation and analysis performed in computer-aided design (CAD) software to calculate the flow of liquids or gases in or around a product.

It is a multiphysics solution because it involves the interaction of several phenomena, including fluid dynamics, thermodynamics, and conservation of momentum. Like finite element analysis (FEA), the fluid volume is divided into smaller elements that are composed into a matrix. CFD has many uses beyond product development and aerodynamics, such as weather forecasting and visual effects.

CFD allows us to design products and systems that meet fluid flow and heat transfer requirements.

Using CFD software, we can calculate:

- the speed and direction of particles inside or outside the model;
- the temperature;
- the pressure;
- vortices, which give an indication of the rotational movement of the fluid at different points.

These results can be calculated and displayed in a model. When displayed in fluid, the results can be represented as contours of different colours, particles, a direction field, or fluid lines. To facilitate the understanding of the behavior and to speed up the calculations, the results can be displayed at a specific cutting plane [6].

The CFD can be executed by following these steps:

1. It starts with a model. Before entering the CFD simulation environment, the 3D CAD part or assembly to be analysed is created. The geometry can be native to the CAD software or imported.

2. The fluid domain is defined, the liquid or gas in the simulation can be either internal, such as water flowing through a piping system, or external, such as air flowing over the exterior surfaces of a vehicle. The volume region is defined and the material properties of the fluid are applied, including density, viscosity, coefficient of thermal expansion, specific heat capacity, and thermal conductivity.

3. Boundary conditions are established. They represent fluid movement at the inlet and outlet of the analysis model. Fluid motion can be defined by flow velocity, inlet and outlet pressure, and mass flow rate. For internal flow, additional boundary conditions include eddy inlet (velocity with both normal and radial components), rotating wall to simulate moving components, and gravity.

4. Thermal conditions are included. Thermal loads in the system can be defined as heat flux, heat flux, convection, and convection radiation.

5. The analysis is carried out.

6. The system is optimized, providing instant feedback to improve the model.

Unsteady and complicated flow can cause pipe and circuit damage due to the velocity of liquid flow.

Pipeline flow is applied in various industries such as chemical industry and petroleum industry, pipeline manufacturing engineering, power plants, biomedical engineering.

In the chemical industry, these problems can occur when transporting oil and gas.

THE ANNALS OF "DUNAREA DE JOS" UNIVERSITY OF GALATI
FASCICLE IX. METALLURGY AND MATERIALS SCIENCE
N°. 4 - 2023, ISSN 2668-4748; e-ISSN 2668-4756
Article DOI: https://doi.org/10.35219/mms.2023.4.07

Phase distribution is an important component in the design of engineering structures, due to its impact on flow rates and pressure rise.

The simulation was carried out in several stages:

The first stage of the simulation was the realization of the objective to be tested and analysed (Fig. 1).

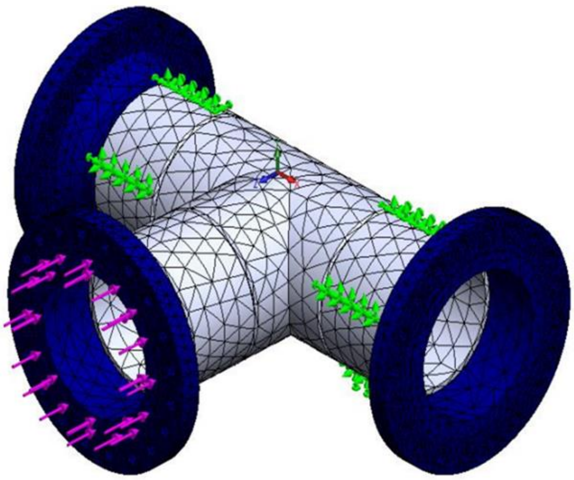

Fig. 1. *Making the model, for analysis and testing*

At this stage, the way it deforms and how the stresses appear, for a T-shaped pipe, was simulated. At the inlet we have a pressure of 10 bar, and the gas is distributed to the other pipes.

I have considered that there are bolts on each flange and the restrictions are as follows:

- each flange is fastened with screws and between the two pipes we have a welding connection.

The second stage of the simulation is presented in (Fig. 2).

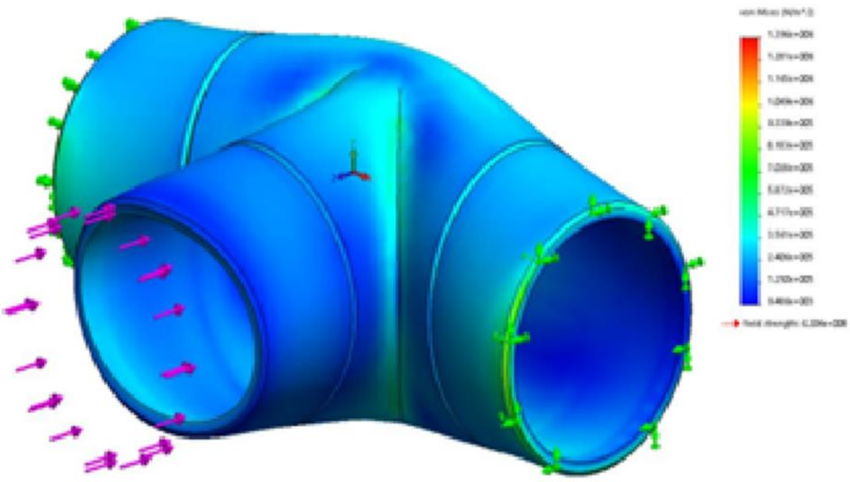

Fig. 2. *Distribution of stresses in pipes*

At this stage we realized the distribution of von Mises stresses in the pipes. I should point out that the model is enlarged several times, to intuitively show the user where deformations occur.

In the third stage of the simulation (Fig. 3) we presented the maximum displacements and their distribution.

THE ANNALS OF "DUNAREA DE JOS" UNIVERSITY OF GALATI
FASCICLE IX. METALLURGY AND MATERIALS SCIENCE
N°. 4 - 2023, ISSN 2668-4748; e-ISSN 2668-4756
Article DOI: https://doi.org/10.35219/mms.2023.4.07

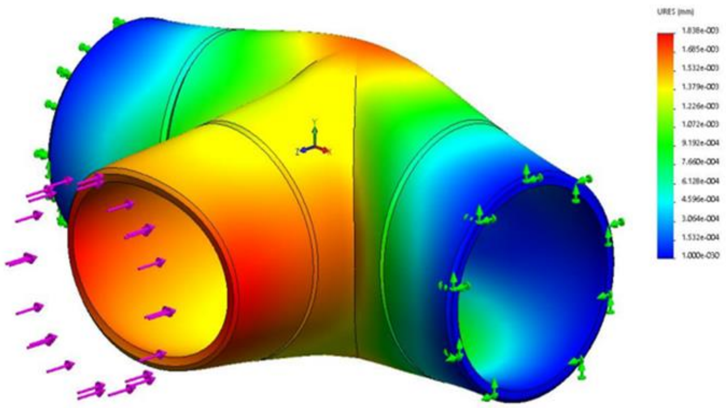

Fig. 3. *Distribution of displacements*

For the maximum displacements and their distribution, it is observed that somewhere at the virtual intersection of the weld lines, a maximum displacement and the displacement is approximately $1.8*10^{-3}$ mm, which is acceptable for the high pressure used in pipeline testing.

The fourth part of the simulation is done on a model of the shape of the part (Fig. 4).

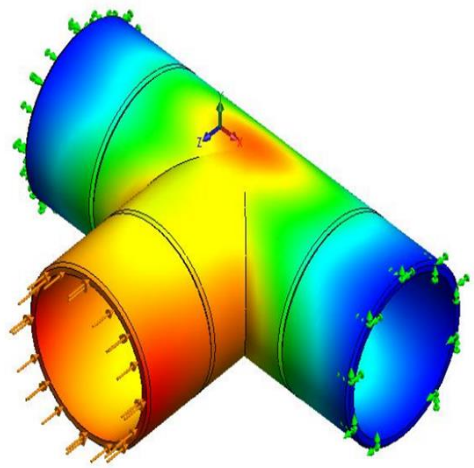

Fig. 4. *Movements without changing the shape of the piece*

3. Conclusions

CFD simulation is important for the design and optimization of structures and systems that interact with fluids.

Using fluid-structure interaction (FSI) modeling and analysis, engineers can optimize the design of structures to reduce the effects of fluid-induced vibration or deformation and improve the performance and safety of these structures.

SOLIDWORKS, allows the simulation of various liquids, gases and oil for various engineering scenarios. Fluid - structure interactions (FSI) refer to the interaction between a fluid flow and a solid structure.

CFD simulation facilitates the analysis of complex fluid flow problems involving liquid-gas, fluid-solid, or fluid-fluid interactions.

An unstable and complicated flow can cause damage to the pipes and the circuit due to the speed of the liquid flow.

The flow of liquids and gases through pipes is applied in various industrial fields, such as the chemical industry and the petroleum industry, pipeline engineering, power plants, biomedical engineering.

References

[1]. Oroveanu T., Stan A., Talle V., *Transportul Petrolului*, Editura Tehnică Bucureşti, 1985.

[2]. Trifan C., *et al.*, *Activitatea Gazieră din România*, Editura Universităţii "Petrol-Gaze" din Ploieşti, 2008.

[3]. Tudorache V. P., *et al.*, *Maintenance of the Romanian National Transportation System of Crude Oil and Natural Gas*, DAAAM, 2013.
[4]. Tudorache V. P., *Research on optimization of the National System of Pipeline Crude Oil Transportation in Romania*, Teza de Doctorat, 2014.
[5]. Ungureanu O., Drug, V., *Transportul Gazelor Naturale*, Editura Tehnică București, 1971.
[6]. Prakash R., Akhtar M. J., Behera R. K., Parida S. K., *Design of a Three Phase Squirrel Cage Induction Motor for Electric Propulsion System*, Third International Conference on Advances in Control and Optimization of Dynamical Systems, Kanpur, India, March 13-15, 2014.