AUTOMOTIVE CFD FUEL TANK FILLING ANALYSIS

Guilherme Wilson Dutra e Duque, guilhermewilson.mecanica@yahoo.com.br
José Antônio da Silva, jant@ufsj.edu.br
Márcio Eduardo Silveira, mesilveira@ufsj.edu.br
UFSJ – Universidade Federal de São João del-Rei

Abstract. *The fuel tanks development process has been modified in the pass of the years. The main function of store the maximum amount of fuel is no more enough to satisfy the consumers. To become useful today, the tanks must occupy the smallest space possible in the whole model volume. Besides, there are lots of studies to preview acoustic problems, volumetric efficiency, less total fill time and other relevant studies. There are some interesting tools to analyze these questions, such the CFD technology, which means "Computational Fluid Dynamics". It can be understood as a software code which solves the fluid or thermodynamics equations by a geometry discretization, into a mesh of finite volumes. On every of these volumes the software get domain results that fit to the other at the neighborhood. The objective of this study is to preview some of these interesting flow effects in the fuel tank filling by using CFD code software. The result may become useful to better understand the whole process and to become possible the troubleshooting even in the project stage. The knowledge applied to this process by using CFD saves costs, improve the final product and give the possibility of deeper analysis, which gets the whole project of the tank to a high accuracy results. Previous studies were realized using meshes between 40,000 and 160,000 volumes, which produced results that fit the reality with good precision. Interesting phenomena could be observed, like sloshing due to filling process. The current work will be realized using improved meshes, with hexahedral volumes in the most part of geometry. There is a deal to turn the cars the smallest and economic as possible in the future, which mean that the car components must be the most efficient as possible. As known, efficiency implies serious studies, which give precision results. The fuel tanks project is no longer simple, and needs more attention. Better and more economic than project and construct lots of possible tank models, is the computer assisted project, which construct and analyze virtual models with wishful precision level in comparison to reality.*

Keywords*: CFD, fuel, filling*

1. INTRODUCTION

The automotive fuel tanks are basically reservoirs made of polymer or metal, with a filler pipe, from where the fuel gets inside the tank, and a vent pipe, which function is to escape the air out during the filling process.

For a long time the only function of the automotive fuel tanks was to contain the fluid inside, and in the past it used to have a reserved place in the remaining car space. With the pass of time, the cars started to become smaller, and the geometry of the fuel reservoir started to fit the remaining empty space at the car. The consequences were that these geometries are different and even complicated on each model.

Complicated geometry implies complicated fuel behavior prediction. There are lots of analyses that become harder with the geometry complication such ways reduce the sloshing, return of fuel at the filler pipe and premature fuel pump disarm.

In attempt of obtain a final project that fits all these requirements, the CFD (Computational Fluid Dynamics) software shows up as a powerful tool do predict the fuel behavior for every kind of geometry, even the most complexes.

Previous works and articles about CFD application at automotive fuel tanks design were developed, such Grogger *et al.* 1998 , Han *et al.*(2002) e Wiesler *et al.(*2002), Greif *et al.* (2003), Ding *et al.* (2005), and Duque et al. (2010). The results obtained at these works revealed that CFD saves invests at the design stage, and get better results with few time cost.

2. OBJECTIVE

The objective of this work is to realize a CFD simulation and analyze the behavior of a tank with considerably complex geometry and show how the CFD code software are capable to predict and treat problems, reducing money spent on experimental process.

3. METHODS

The first thing to simulate the problem is to get a CAD model of the analyzed fluid domain. Even simulating one tank, the walls and solid components aren't relevant to the results. It's important to represent correctly the fluid domain that will be analyzed.

To get a good fluid representation, the CAD geometry must be discretized in smaller geometry pieces. This large group of pieces is called mesh, generated by specific software or a CFD integrated one. The amount of mesh volumes is defined by the computer resources available, and the refined area needs. It's called finite volume finite analysis.

One time the mesh is well defined, it will be inserted to software which function is to define the boundary conditions. This step of simulation is very important, because is when all physic process will be modeled. After this step, solver software will get the results by iteration process, until the converged to a specific RMS level. The results are then post-processed into a graphic interface to show clearly the fluids behavior.

At this simulation, one general geometry tank similar to industry ones was designed at CAD software, and exported to the CFD Software in a common extension type called IGES (Fig. 1).

After this step, the mesh was generated, with tetrahedral volumes to fit the entire geometry properly. The final mesh was composed by about 160 thousand finite volumes, which was enough to represent the fuelling process, accordingly to the available computer resources.

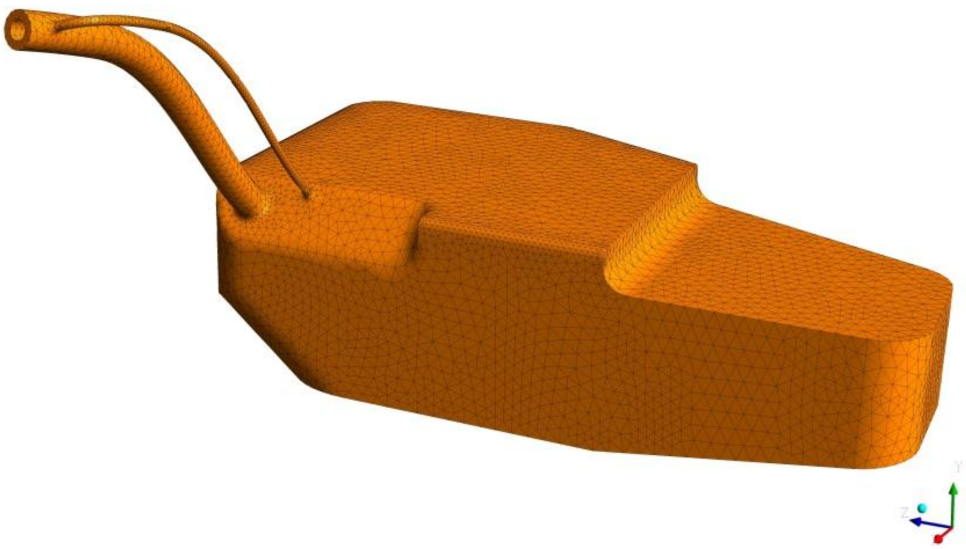

Figure 1. Tank CAD already discretized into tetrahedral mesh

The solution of the problem was made by a code (*CFX*) that resolves all of the flow equations based on the principle of conservation of mass, momentum and energy by discrete methods that identify the region of interest. Some characteristics are relevant to be explained.

3.1. Free Surface Model

When simulating fluid interaction, there are two types of situation: mixture or free surface. In this work, the behavior of air in contact to fuel generates one well defined free surface. The CFD code used understands this behavior through the eulerian-eulerian multiphase model.

3.2. Interfacial Area Density

The interfacial transfer of momentum, heat and mass depends directly on the surface area of contact between the two phases. This is characterized by the interfacial area per unit volume between phases α and β, known as interfacial area density, $A_{\alpha\beta}$.

As the model used is the free surface and are only considered air and fuel phases, the Eq. (1) is used to calculate the density of interfacial area.

$$A_{\alpha\beta} = |\nabla r_\alpha| \tag{1}$$

3.3. Hydrodynamic Equations Applied to Heterogeneous Model

3.3.1. Momentum Equation

$$\frac{\partial}{\partial t}\left(r_\alpha \rho_\alpha \boldsymbol{U}_\alpha\right) + \nabla \cdot \left(r_\alpha\left(\rho_\alpha \boldsymbol{U}_\alpha \times \boldsymbol{U}_\alpha\right)\right) = -r_\alpha \nabla p_\alpha + \nabla \cdot \left(r_\alpha \mu_\alpha\left(\nabla \boldsymbol{U}_\alpha + (\nabla \boldsymbol{U}_\alpha)^T\right)\right) + \sum_{\beta=1}^{N_p}\left(\Gamma_{\alpha\beta}^{+}\boldsymbol{U}_\beta - \Gamma_{\beta\alpha}^{+}\boldsymbol{U}_\alpha\right) + S_{M\alpha} + M_\alpha \quad (2)$$

$S_{M\alpha}$ describes the momentum sources due to external forces to the body, and the term $\left(\Gamma_{\alpha\beta}^{+}U_\beta - \Gamma_{\beta\alpha}^{+}U_\alpha\right)$ represents the momentum transfer induced by mass transfer at the interface.

3.3.2. Continuity Equation

$$\frac{\partial}{\partial t}\left(r_\alpha \rho_\alpha\right) + \nabla \cdot \left(r_\alpha \rho_\alpha U_\alpha\right) = S_{MS\alpha} + \sum_{\beta=1}^{N_p} \Gamma_{\alpha\beta} \quad (3)$$

The term $S_{MS\alpha}$ describes the sources of specified mass, and $\Gamma_{\alpha\beta}$ is the mass flow rate per volume unit from phase β to phase α. This term is only used when mass transfer occurs at the interface.

3.3.3. Volume Conservation Equation

The volume fractions must sum to a unit in accordance with the Eq. (4).

$$\sum_{\alpha=1}^{N_p} r_\alpha = 1 \quad (4)$$

This equation can be combined with the continuity equation of each phase to obtain the conservation equation of the volume transported, which leads to the Eq. (5).

$$\sum_\alpha \frac{1}{\rho_\alpha}\left(\frac{\partial}{\partial t}(r_\alpha \rho_\alpha) + \nabla \cdot (r_\alpha \rho_\alpha U_\alpha)\right) = \sum_\alpha \frac{1}{\rho_\alpha}\left(S_{MS\alpha} + \sum_{\beta=1}^{N_p} \Gamma_{\alpha\beta}\right) \quad (5)$$

3.3.4. Pressure composition

The entire hydrodynamic equations set represent four of N_p+1 equations in five $N_{p:}$ $U_\propto, V_\propto, W_\propto, r_\propto, p_\propto$, which are unknown. N_p-1 equations are required to close the system, which can be obtained through the composition of pressure, considering that all phases share the same pressure field: P_α=P for all α=1,...,N_p.

3.4. Simulation Type

The simulation was transient, with total time of 70s, time step of 0.25s. For each step were taken 50 iterations to achieve the necessary convergence. The simulation total time refers to the approximated time necessary to fill up the entire tank, based on fuel flow defined and the tank geometry volume.

3.5. Domain Definition

The domain used was biphasic heterogeneous, and the analyzed fluids were atmospheric air and liquid gasoline, with a gravity of 9.81 m/s² and reference pressure of 1 atm. There is a feeding pipe of 50 mm diameter, 25 mm filler output, and a 20 mm diameter vent pipe. Thermal analysis was not performed, since the goal was simply to analyze the flow at a temperature of 25 °C. The turbulence model used was the SST (Shear Stress Transport), suitable for analysis of both the interaction with the same fluid as the tank walls interaction. The tank is initially drained, at atmospheric pressure and temperature of 25 °C.

3.6. Boundary conditions

3.6.1. Entry

The gasoline enters the feeding pipe with a mass flow of 0.703 kg / s, corresponding to 1 liter per second.

3.6.2. Output

The tank output is represented (Fig.2) as the empty space defined by the junction of the vent pipe to inside the feeding pipe. The hole represents the extruded pump filler placed into the feeding tube. The output is defined as an opening where fluid flow can occur in both directions. This opening is represented as an annular surface, perpendicular to flow entrance.

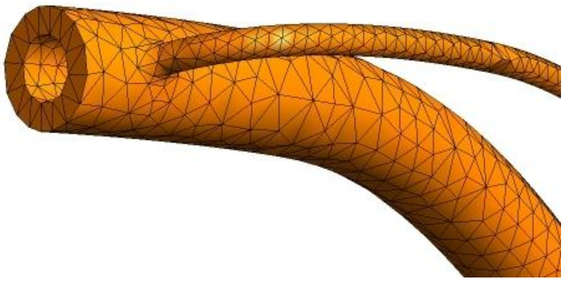

Figure 2. Detail of the output system

3.6.3. Startup

At first it admits a full tank of air modeled as ideal gas at 1 atm and with zero velocity for the three coordinates X, Y and Z.

4. RESULTS

This kind of simulation returns a lot of interesting analysis such pressure fields, fluid turbulence, free surface observation, sloshing due to filling process, fuel return in feeding pipe, and others.

The results are returned as graphic representation to easy visualization, such as process comprehension. The 3D geometry at the input is now represented in wireframe, to show the properties variation at the boundary such inside the fluid by color scales. This representation is generated at the post processing software part by interpolation between each one of the mesh nodes.

To this simulation is special, some relevant results will be shown, such sloshing due to filling process, fuel return at feeding pipe and vent pipe, pressure fields and free surface representation.

4.1. Free Surface Visualization

The fuel contact with air inside the tank generates a free surface, and it can display the fuel level, such as other visual behavior (Fig. 3). The mesh resolution cause direct influence over the free surface details, so it must be concerned at the simulation definition. The finer the mesh, the more realistic is the free surface, although to much volumes at mesh require a lot of computational resources.

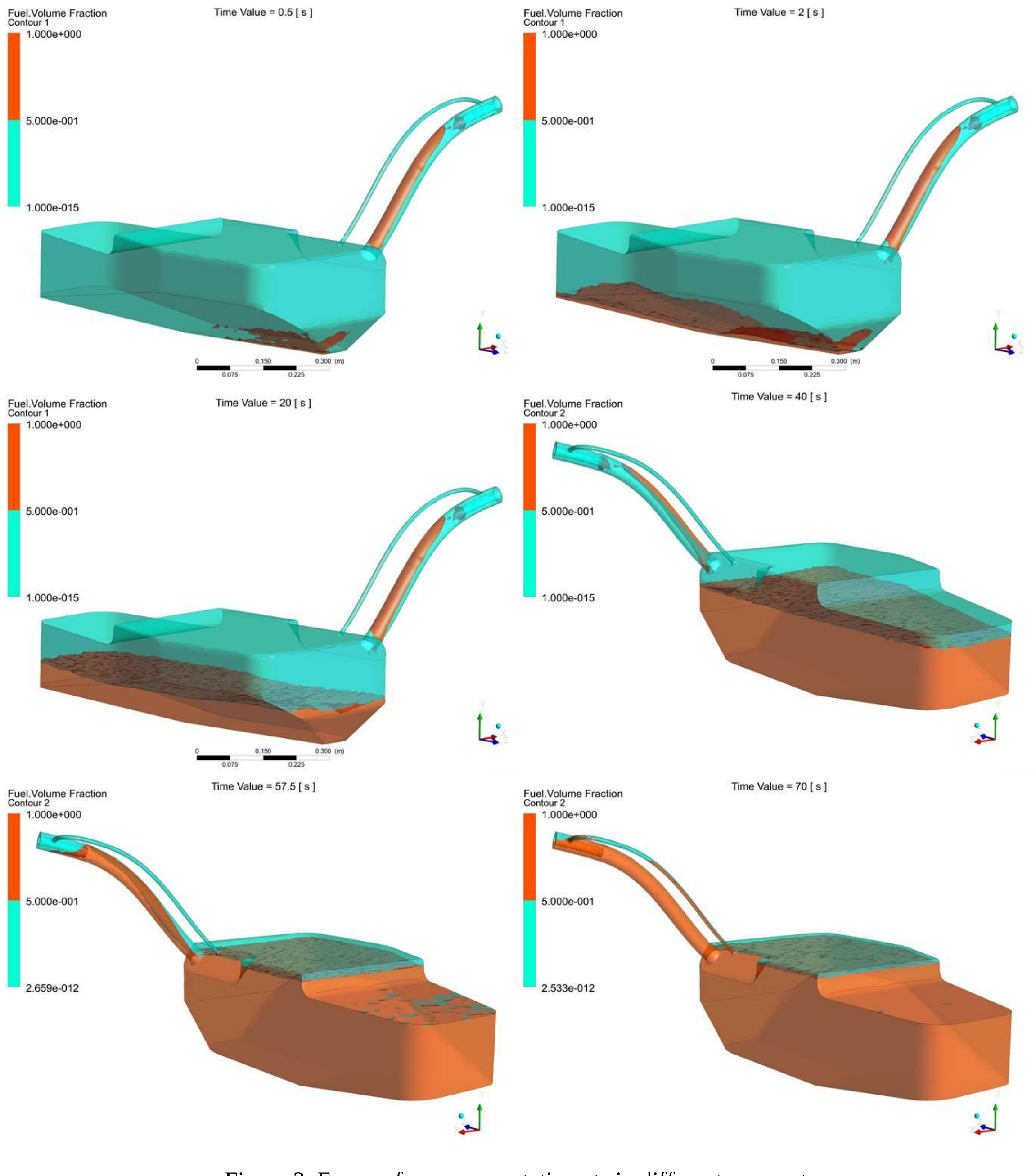

Figure 3. Free surface representation at six different moments

The fluid seems to disappear between the end of feeding pipe and the free surface, what can be explained to ripple effect. The relatively small mesh number of volumes defined doesn't allow observing the free surface at this situation, but it doesn't relevantly affect the expected results.

At 70 seconds of simulation, it's possible to notice that the feeding pipe and either the vent pipe are filled with fuel, in spite of the tank is still not completely filled.

4.2. Sloshing Due to Filling Process

When air and fuel remains static, the free surface is plane, but the fuel flow to inside the tank, generates lots of distortion around the whole surface, and leads to air bubbles and fuel waves. By taking a section plan, transversal to filling pipe, it's possible to analyze the effects caused by fluids revolving (Fig. 4).

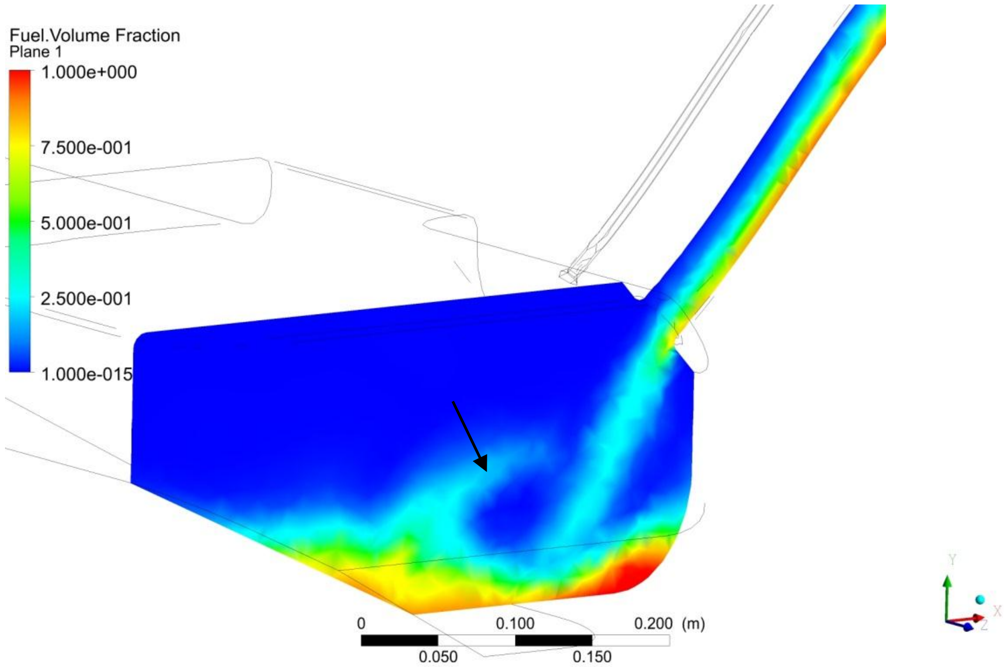

Figure 4. Revolving fuel at the tank bottom, due to sloshing

4.4. Undesirable Fuel Returning

Unfortunately, some fuel can return at the pipes before filling process is completed, and as the pump filler disarms by fuel contact, these returns must be avoided. The first thing to do is understand how the returns occur. One way to obtain this is by pressure field's observation (Fig. 6), or by free surface behavior analysis (Fig. 5).

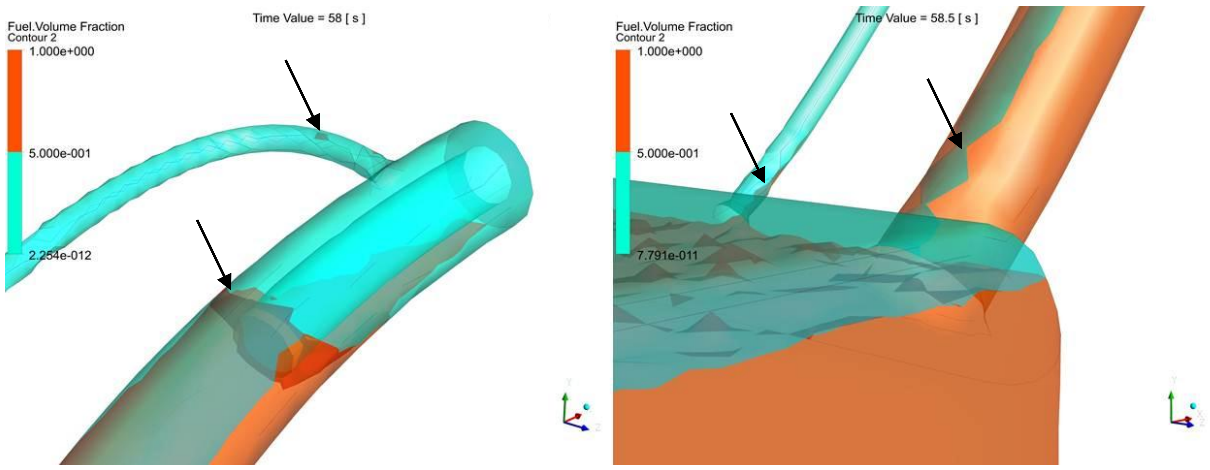

Figure 5. Undesirable returning fuel at pipes, before filling completion.

The fuel starts to return at the pipes by the time of 58 seconds of simulation. It's possible to verify small amounts of liquid inside the vent pipe (Fig. 5), even when the tank wasn't completely filled. It implies in volumetric inefficiency.

4.3. Pressure Fields

At the filling process, some air got prisoned at tank's top, due to over atmospheric pressure, which leads to returning fuel at filling and vent pipes. When at design stage, this kind of simulation can easily show where the air becomes prisoned, by plotting the pressure fields in color scale (Fig. 6).

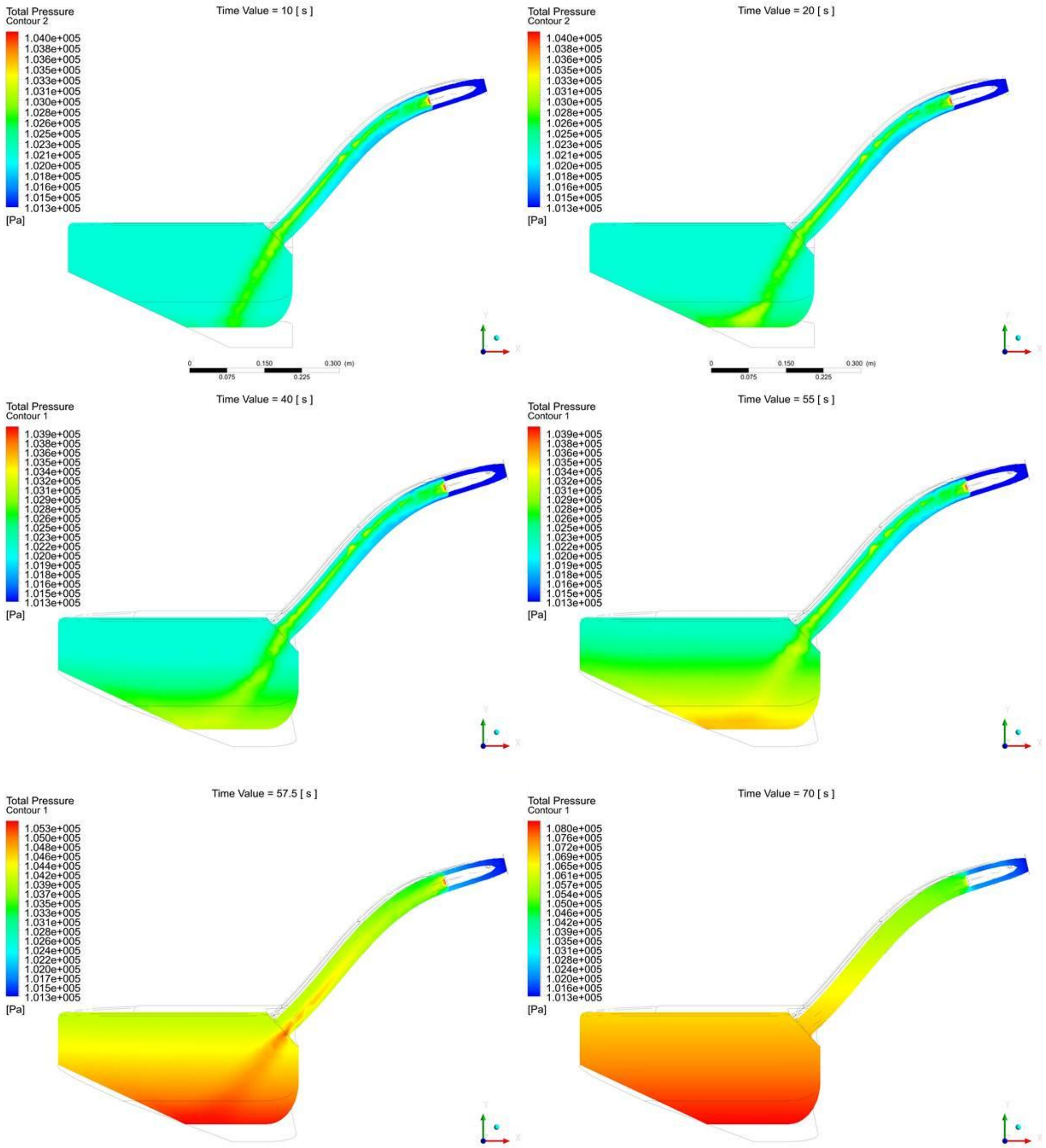

Figure 6. Pressure fields represented at six different moments.

The pressure elevates approximately 7 kPa during the entire filling process, doing the fuel to return previously.

4. CONCLUSION

The CFD simulation revealed itself as efficient way to diagnose problems at automotive fuel tanks, even at design stage. The use of computational resources can save time and money in developing fuel tanks and other components. The simulation in this study managed to express the desired results even for a still small number of finite volumes in the mesh. The choice of a correct number of volumes should be based on available resources, and varies according to the requirements of the simulation. It is concluded that computational fluid dynamics, or CFD, perfectly fits the project needs an automotive fuel tank.

3. ACKNOWLEDGEMENTS

The team appreciates the partnership with the professor Luben Cabezas Gomez (PUC-MG) by the transfer of computing resources to perform such work.

4. REFERENCES

Duque, G. W. D. e, Silva, J. A. da, Silveira, M. E., Simulação de enchimento de um tanque de combustível automotivo utilizando técnicas de CFD, *2010 Nono Simpósio de Mecânica Computacional, (IX SIMMEC).*

Ding, P., Buijk, A. J., van der Vee, W. A., 2005. Simulation of fuel tank filling using a multi-material euler solver with multiple adaptive domains. *2005 SAE International.*

Greif, D., Wiesler, B., Alajbegovic, A. , 2003. Two-phase tank filling simulation of an automobile tank system. *Computational Fluid and Solid Mechanics*, pp. 919 - 922.

Grogger, H.,Philipp, H.,Alajbegovic, A., Kolbinger, H.-J., Nurna, P., 1998.Two-phase flow calculation of the tank filling. *JSAE Spring Convention, Yokohama*. Japan, May 20-22.

Han, J., Alajbegovic, A., 2002. Simulation of multiphase flows in complex geometry using a hybrid method combining the multi-fluid and the volume of fluid (VOF) approaches. *ASME FEDSM*. 02 - 31153, Montreal.

Wiesler, B., Greif, D., Alajbegovic, A., 2002. Recent progress in tank-filling simulation. *VDI Würzburg*. October, 1-2.

5. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.