

ENCIT-2018-0692 HEAD LOSS OPTIMIZATION

Vinícius Lopes Nogueira

Diego Fauser

Rogério Gonçalves dos Santos

Faculdade de Engenharia Mecânica da Universidade Estadual de Campinas – FEM/UNICAMP

Rua Mendeleiev, 200 – CEP 13083-860

Cidade Universitária “Zeferino Vaz” Barão Geraldo

Campinas - SP

vinicius.lnogueira@gmail.com, diegofauser@gmail.com, roger7@fem.unicamp.br

Abstract. *The purpose of this study is to create a model for topological optimization of geometries to reduce head loss caused by fluid recirculation. The main objective is to better handle difficulties found when designing inlet manifolds of internal combustion engines. The problem is analyzed in two dimensions using a finite volume code and solving momentum and mass equations. In each iteration, a new fluid speed profile is calculated, these are used to define fluid recirculation zones, these zones are then defined as solid, thus, preventing fluid circulation and reducing head loss, and ultimately, optimizing the geometry. The resulting optimized geometry showed a quite significant reduction in head loss, approximately 8%.*

Keywords: *Airflow, Head loss, Optimization*

1. INTRODUCTION

There is a great challenge when designing components with internal air flow, these components tend to have complex geometry in order to minimize head loss, some examples of that include: air conditioning systems, exhaust systems, inlet manifolds for internal combustion engines, etc. Inlet manifolds are responsible for supplying air or an air/fuel mixture to the cylinders of the engine. In an ideal scenario, the air quantity in each cylinder is the same - guaranteeing a better fuel usage, and consequently a higher efficiency - and head loss is kept to a minimum, however, in reality, limited space and complex geometries contribute to a significant head loss during this process.

An internal combustion engine's torque is an increasing function of the quantity of air admitted inside its cylinders, which means that volumetric efficiency is extremely important, some engines, aiming to increase volumetric efficiency even incorporate turbochargers or mechanical compressors (Wyszynski *et al.* 2002). Thus, optimizing the relevant geometries to conduct airflow to lower head loss could improve engine efficiency considerably.

According to Stockel *et al.* (1999), the total efficiency of a vehicle is around 15% in relation to the energy generated by the fuel, while 85% are lost through heat, friction, etc. As mentioned before, the existing components in an inlet manifold are complex, making necessary the use of computational analysis to fully optimize their performance (Lira, Carvalho, Costa, & Henríquez, 2017). Considering the fact that studying simulation in inlet manifolds requires understanding of complex geometries with internal fluid flow, topological optimization offers advantages over conventional optimization techniques, since the method allows shaping the geometry according to the proposed fluid flow. (Stephan, Häussler, & Böhm, 2009).

Topology optimization is a computational approach to synthesize structures without predefined shapes. This freedom makes topology optimization a great alternative to find new, high-performance structural designs (Liu & Tovar, 2014). Therefore, this research aims to find a better geometry, minimize head loss, and subsequently improve – through topological optimization – the efficiency of systems that have the same design complications.

2. METHODOLOGY

The standard geometry analyzed consists of a rectangular box with one inlet and one outlet. Boundary conditions are specified for every node located in the boundaries of the domain and are constant throughout the simulation. The specified properties are: temperature, horizontal and vertical velocity of air flow, pressure and solid matrix.

A criterion for maximum recirculation is established so that an evaluation can be made if and specific region (control volume) should be considered a solid wall, not allowing fluid flow.

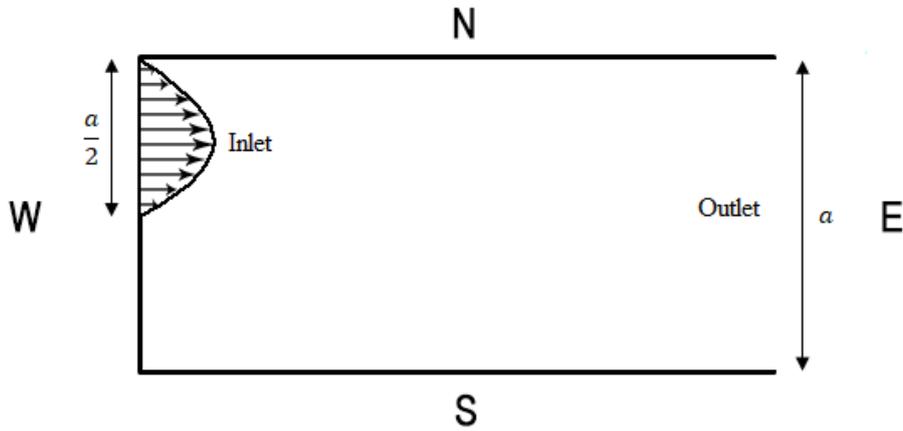


Figure 1. Simplified Geometry

Analyzing the simplified geometry (Fig. 1), the north and south boundaries do not allow mass flow, thus, horizontal and vertical velocities are null. At east and west there are two different regions, one where the solid matrix is defined as existent (value set to 1) and will not allow fluid flow, and a second one, where there is mass flow (inlet and outlet), and thus, the solid matrix is inexistent (its value is set to zero). This numerical method is based on the work done by Armengol (2015) that utilizes a matrix to characterize the nature of the control volume, allowing for calculations about properties and coefficients to be made and solutions to the governing equations to be solved in the whole domain.

Inlet horizontal velocity is an input parameter and its value in each inlet point can be set by

$$u = -4u_{max} \left(\frac{y^2}{a^2} - \frac{y}{a} \right) \quad (1)$$

As a simplified way to define recirculation zones, every grid node that returns a negative value for horizontal velocity ($u < 0$), will be considered above the recirculation criteria and will be treated as a solid, preventing fluid flow through that area.

The properties attributed to the geometry are: length = 0,6 m, and width (a) = 0,05 m. The grid used has 301 nodes in the X axis and 151 in the Y axis. Air viscosity and density are considered constant and are taken at 25°C, $\mu = 1,84E-5$ N.s/m² and $\rho = 1,19$ kg/m³ (Fox, McDonald, Pritchard, & Leylegian, 2011). The flow in the inlet is considered fully developed and the speed profile is defined through Eq. 1 where $u_{max} = 0,1$ m/s.

Due to the bottleneck inlet, most likely big losses will be seen right underneath this region, and the air flow will divert from its original path, a recirculation zone will appear in this exact area, resulting in head loss, this effect is shown in Figure 2.

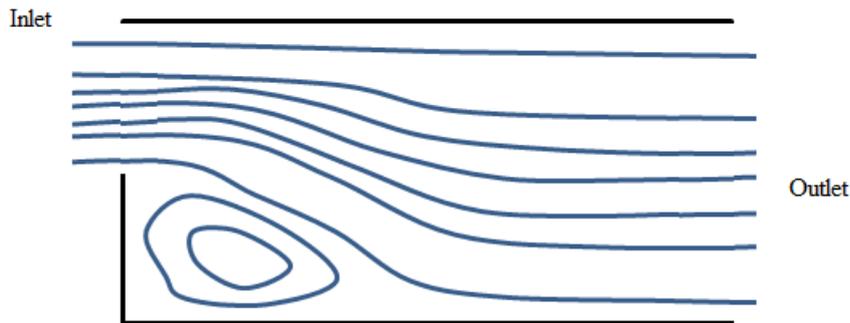


Figure 2. Recirculation zone

This region has great room for improvement and optimization, it's expected that the code will identify the negative values for horizontal velocity in the area and create a new solid matrix after each iteration throughout that whole recirculation zone, reducing head loss.

The code used is set to solve momentum and mass equations and an equation to solve head loss (derived from the energy equation). The equations presented below are the mass and momentum equations for a control volume in two dimensions.

$$\begin{cases} \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \\ \frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = -\frac{1}{\rho} \frac{\partial P}{\partial x} + \nu \left[\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right] \\ \frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} = -\frac{1}{\rho} \frac{\partial P}{\partial y} + \nu \left[\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right] \end{cases}$$

The head loss is calculated from the following equation, which is a derived form of the energy equation.

$$h_{it} = \frac{1}{\dot{m}} \left[\int_{sc} V^2 \cdot \rho \vec{V} \cdot d\vec{A} + \int_{sc} g_z \cdot \rho \vec{V} \cdot d\vec{A} + \int_{sc} \frac{p}{\rho} \cdot \rho \vec{V} \cdot d\vec{A} \right]$$

3. RESULTS

3.1 Standard Geometry

A simulation was conducted on the initial, standard geometry; this result will serve as a benchmark to verify the difference in head loss in between the two cases. The speed profile graph (Fig. 3) shows negative horizontal velocity below the entrance, which normally characterizes a recirculation.



Figure 3. Benchmark speed profile

After reaching the stopping criteria, the head loss found for the benchmark problem was $h_{it} = 2,549\text{E-}03 \text{ m}^2/\text{s}^2$.

3.2 Optimized Geometry

New simulations show results for a geometry that has a growing solid frame where the recirculation zone was previously found

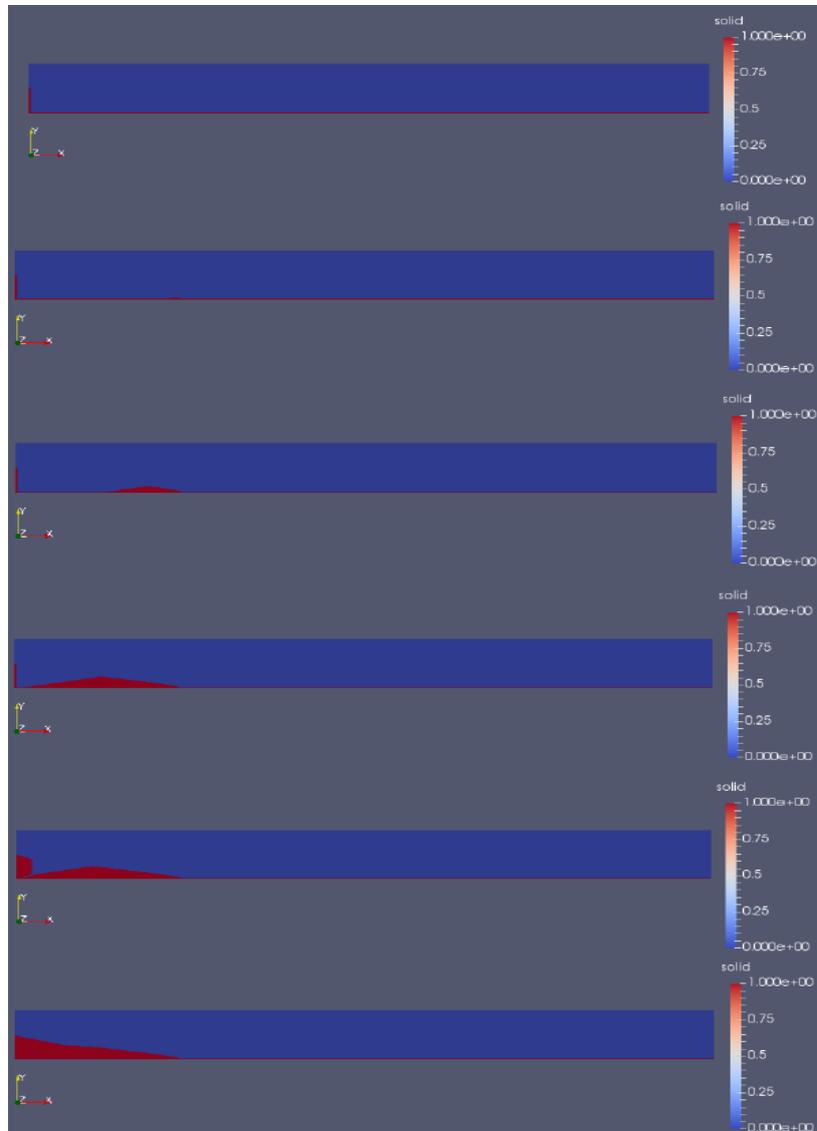


Figure 4. Evolution of solid matrix throughout the simulation

Figure 4 illustrates six stages in the increasing size of the solid mass underneath the fluid entrance.

Comparing the results obtained from the benchmark simulation and the optimized geometry (Fig. 5), the area beneath the entrance shows better results for horizontal velocity.

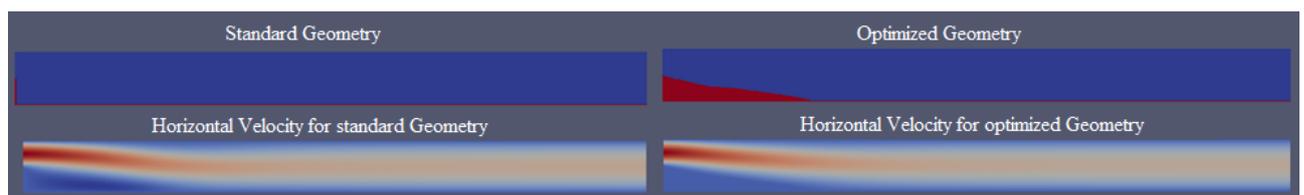


Figure 5. Comparison between benchmark and optimized problem

The results of head loss calculated for the optimized geometry was $h_{lt} = 2,349E-03 \text{ m}^2/\text{s}^2$, a reduction of 7,85% when compared to the results in the standard geometry (around $2,549E-03 \text{ m}^2/\text{s}^2$).

4. CONCLUSIONS

The code used was capable to identify the recirculation zone, and replace it with a solid wall, limiting fluid flow through those zones. The flow velocity confirms the existence of a recirculation zone by showing negative horizontal speeds underneath the restricted entrance, it also shows an improvement in the fluid flow when that zone is removed from the path followed by the fluid.

Lastly, the head loss found for the standard geometry was $2,549\text{E-}03 \text{ m}^2/\text{s}^2$, while the one found for the new geometry was $2,349\text{E-}03 \text{ m}^2/\text{s}^2$, these numbers show a reduction of 7,85% in head loss, confirming a better performance for the optimized geometry.

5. REFERENCES

- Armengol, J. M. (2015). *Estudo Numérico de Crescimento de Gelo Poroso Entre Placas Planas Paralelas*. Campinas: Unicamp.
- Fox, R. W., McDonald, A. T., Pritchard, P. J., & Leylegian, J. C. (2011). *Introduction to Fluid Mechanics, 8th ed.* Hoboken, NJ (US): John Wiley & Sons, Inc.
- Lira, J. d., Carvalho, R. d., Costa, J. d., & Henríquez, J. (2017). *Análise de um Coletor de Admissão para Motor Veicular 1.0 pelo Método de Dinâmica dos Fluídos Computacional*. Recife: Universidade Federal de Pernambuco.
- Stephan, D.-I. M., Häussler, D.-I. D.-P., & Böhm, D.-M. M. (2009). *CFD Topology Optimization of Automotive Components*. Karlsruhe, Alemanha: FE-DESIGN GmbH.
- STOCKEL, M. W. (1999). *Auto Fundamentals: how and why of the design, construction, and operation of automobiles: applicable to all makes of and models 3ed.* Goodheart-Wilcox Publisher.
- WYSZYNSKI, L. P.; STONE, R.; KALGHATGi, G.T. *The volumetric efficiency of direct and port injection gasoline engines with different fuels* In : WORLD CONGRESS & EXHIBITION TECHNICAL PAPERS, 2002, Detroit. Proceedings... Detroit, University of Oxford and Shell Global Solution 2002. 2002-01-0839
- Liu, K. & Tovar, A. *Struct Multidisc Optim* (2014) 50: 1175. <<https://doi.org/10.1007/s00158-014-1107-x>>

6. RESPONSIBILITY NOTICE

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