International Journal of Engineering &Technology, *8 (3) (2019) 337-342*

International Journal of Engineering & Technology

Website: www.sciencepubco.com/index.php/IJET

Research paper

Computational validation of the design of an air-biogas mixer for a turboalimented diesel engine

Javier Carpintero Durango [1] *, Jonathan Fábregas Villegas [2], Henry Santamaría De La Cruz [2], Saul Pérez Pérez [2], Guillermo Valencia Ochoa [3]

[1] Department of Civil and Environmental, Universidad De La Costa, Calle 58 #55-66, 080002 Barranquilla, Atlántico, Colombia
[2] GIIMA – IMTEF – GIIM, Research Group, Faculty of Engineering, Universidad Autónoma del Caribe, Colombia
[3] Kaí Research Group, Faculty of Engineering, Universidad del Atlántico, Colombia
**Corresponding author E-mail: jcarpint3@cuc.edu.co*

Abstract

The design of two air-biogas mixers according to the design literature of these devices for Diesel engine applications was studied through computational fluid dynamics (CFD). The gradient of hydrodynamic variables was profiled on the longitudinal axis of the compared equipment. This research presents a comparison instrument about the efficiency of the design methodology for sizing gaseous fuel air mixing equipment. Comparing the design methodologies of Agudelo and Mitzlaff mixing tubes, the profiles of velocities, pressure profiles and volumetric fraction profiles of the mixtures were obtained.

Keywords: *Spacer Ring; Biogas; Diffuser Cone; CFD (Computational Fluids Dynamics); Multicomponent Flow.*

1. Introduction

Computational Fluid Dynamics (CFD) is a method that solves through numerical analysis, the governing equations of substances with continuous deformation under the action of shear stress [1]. Thus, it can be considered as an alternative to predict responses of complex models of continuity, momentum and energy that are not possible to solve by analytical procedures.

The proper implementation of the CFD can describe the behavior of fluids in engineering flow devices. Biogas is a biofuel that has the potential to work with diesel engines and reduce polluting emissions. However, the performance of these energy integrations depends on the optimal design of the air-biogas mixer. The most accepted analytical design methodologies of this air-biogas mixer is the one proposed by Von Mitzlaff [2] and Agudelo & Mejía [3]. There are qualitative comparisons of these approaches by F. Bermejo & W. Orozco [4][5]. From the qualitative point of view, these authors suggest that the approach of Agudelo and Mejía is more complete than the one proposed by Von Mitzlaff. However, the differentiation of these techniques can be better based on the interpretation of hydrodynamic variables of the equipment, such as pressure, concentration or velocity gradients. Therefore, the techniques of dynamic modeling and simulations by finite elements are very useful methods that allow to reduce the costs of experimentation and working time with conservative results. Researchers like J. Aditya & K. Umesh [6] analyzed the effect of mixing a carburetor system, measuring its efficiency through CFD simulations of the air-gas mixture. It was possible to validate the model with an experimental camera and in this way the results of the CFD method. Besides, S. Biradar et al. [7] developed the optimization of the mixed air-gas process for an engine, by means of simulations by CFD, obtaining as results a good mixing efficiency based on the detailed analysis of the flow and the geometric design of the system.

Consequently, K. Peda et al. [8] carried out a CFD study to compare the characteristics of the air-fuel mixture, taking into account the direction of the injector and the configurations of the simulation, obtaining as a result of the CFD that the aspiration using a venturi with two orifices provides an adequate mixture of air and fuel. In other investigations like those of Sanket & H. Shinde [9] where the importance of the production of more environmentally friendly fuels is highlighted and they propose a method for the generation of biogas through anaerobic fermentation. while Fajri Vidian et al. [10] evaluated the behavior of a burner system that uses methane working gas through simulation by CFD, obtaining detailed analysis of the preheating zone and combustion zone in the device. In this study, the CFD is applied to numerically solve the conservation equations that govern the mixing process of air and biogas from mixers, to estimate which analytical design methodologies can be accepted, taking into account turbulence models and auxiliary equations in the design.

2. Theory and formula

The development and application of the associated phenomena, with the behavior of microchannel technology, are described by their formulation of mathematical models, as explained below.

2.1. Computational model

Fig. 1 shows the results of a CAD software related to the use of the dimensions obtained by the design methodologies of Agudelo and Mitzlaff to analyze biogas-air mixtures through CFD. Subsequently, the design approaches are analyzed with the same boundary conditions to evaluate which of the design proposals has better patterns in throat speed, the kinetic energy of turbulence, concentration gradients, among others.

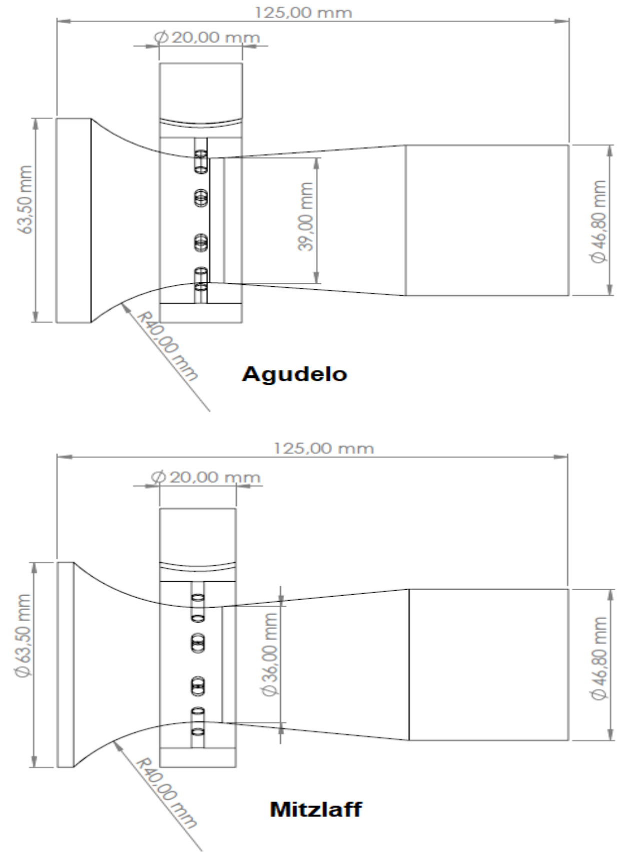


Fig. 1: Dimensions of the Designs - Agudelo and Mitzlaff Methodologies.

Then, the fluid volume models (VOF) are activated to pattern two or more immiscible fluids while solving a single set of impulse equations. In addition, the volume fraction of each of the fluids in the whole domain is tracked and the turbulence model k-ε represented in equations (1-3) is used, taking into account the transport equations for turbulence kinetic energy and its dissipation rate. Under these assumptions, the boundary conditions are associated to solve the modeling as shown in fig. 2. In this case, two entries of chemical species are identified, that is, one for air and the other as biogas (proportions of 40% of carbon dioxide and 60% methane) for both design methodologies. The same hydraulic conditions are used in order to evaluate their behavior.

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}(\rho k u_j) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\partial_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + G_b - \rho\varepsilon - Y_M + S_k \quad (1)$$

$$\frac{\partial}{\partial t}(\rho\varepsilon) + \frac{\partial}{\partial x_j}(\rho k u_j) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial s}{\partial_j}\right] + \rho C_1 S\varepsilon - \rho C_2 \frac{\varepsilon^2}{k + \sqrt{v\varepsilon}} + C_{1\varepsilon}\frac{\varepsilon}{k}C_{3\varepsilon}G_b + S_\varepsilon \quad (2)$$

$$C_1 = max\left[0.43, \frac{n}{n+5}\right], n = S^{\frac{k}{\varepsilon}}, S = \sqrt{2S_{ij}S_{ij}} \quad (3)$$

Where G_k represents the generation of turbulence due to the average velocity gradients, G_b is the generation of energy by buoyancy and Y_M is the fluctuating expansion in the turbulence.

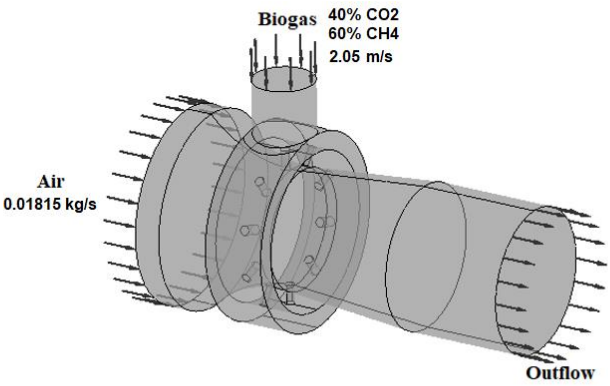


Fig. 2: Conditions for Mixing Systems.

3. Results discussions

Fig. 3 shows the air distribution within the mixer modeled by the methodologies compared in this study. It is observed that the air distribution in the Mitzlaff methodology tends to homogenize the biogas in greater proportion on the part of the divergence of the mixer with respect to the design proposal of Agudelo.

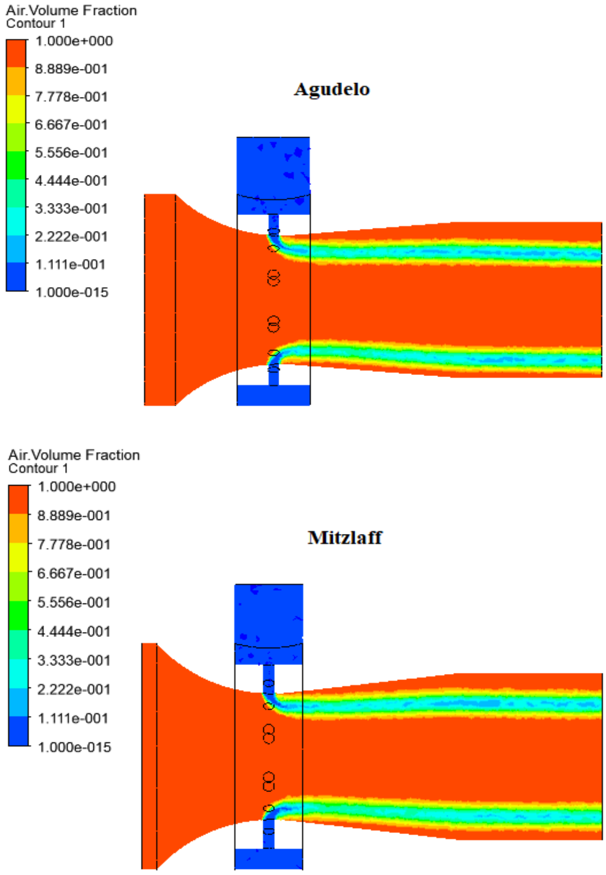


Fig. 3: Air Concentration Contour.

Fig. 4 presents the pressure contours of each design proposal. It was observed that the Mitzlaff methodology tends to produce greater pressure drops in the throat of the mixer, compared with the design methodology of Agudelo. That is, below 50Pa in the region of greatest pressure drop for both mixer design strategies according to Fig. 5.

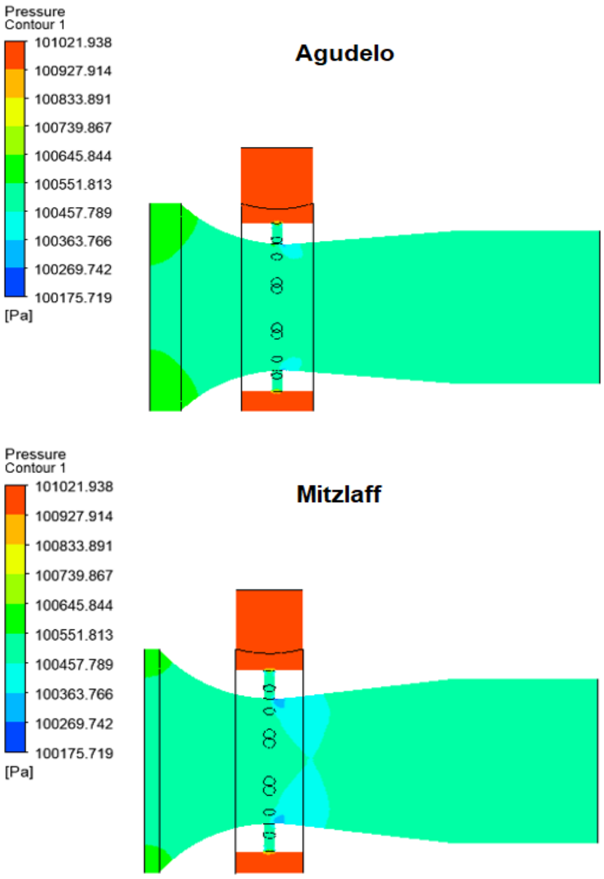


Fig. 4: Pressure Contour of Design Methodologies Evaluated.

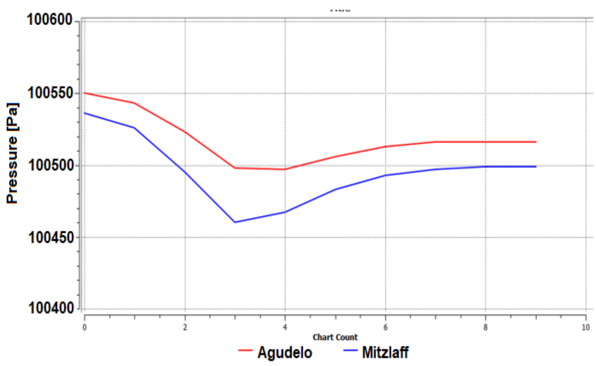
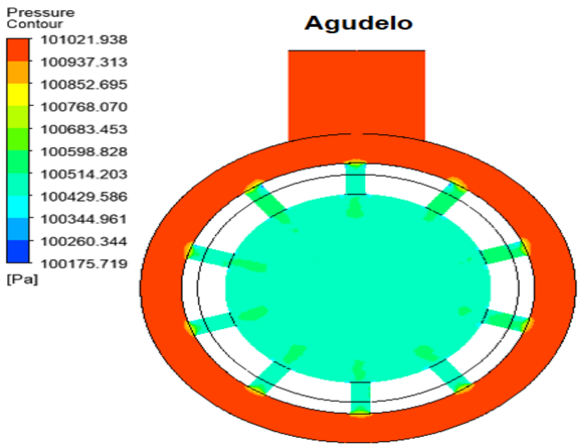


Fig. 5: Pressure Along the Longitudinal Axis of Mixers Evaluated.

Fig. 6. Presents a sectioned view of the fluid contained in the mixer with the distribution of biogas pressures in the mixer. It is observed that the approach of Agudelo induces the biogas to be injected to the throat of the mixer at pressures above 253 Pa with respect to the Mitzlaff strategy.



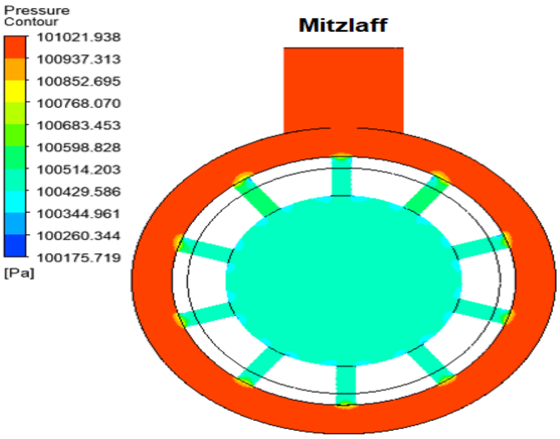


Fig. 6: Pressure Along the Longitudinal Axis of Mixers Evaluated.

Because of this, Fig. 7 reveals that the air surface velocity in the Mitzlaff methodology tends to be higher in the throat of the mixer than that detected with Agudelo proposal. Fig. 8 details that in the throat this speed is up to 2m / s higher in the Mitzlaff methodology than the strategy of sizing air-biogas mixers by Agudelo. Fig. 9 is associated to a plane of the mixer designed by both methodologies and it is observed that the gradients of the surface velocity of the air in the case of Mitzlaff are greater with respect to the radius of the throat, than when compared with that of Agudelo.

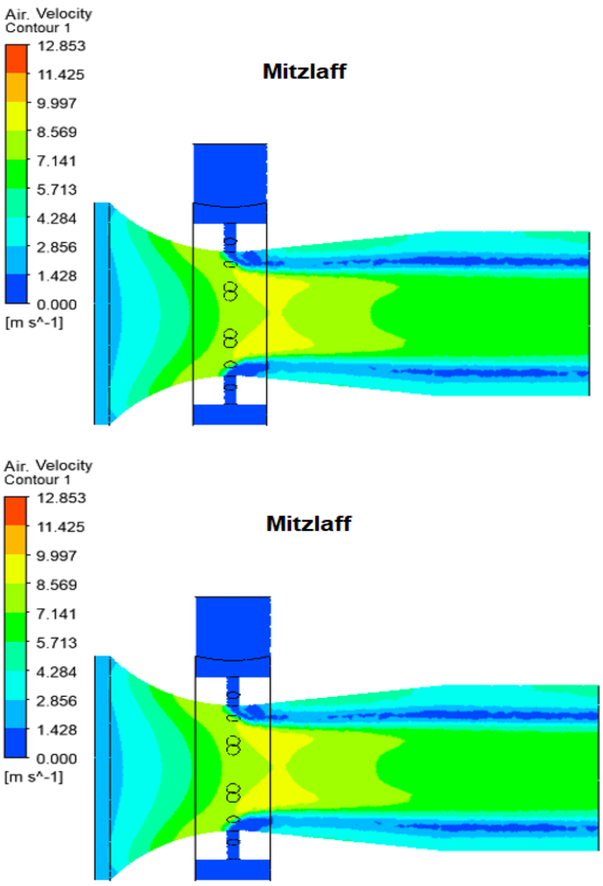


Fig. 7: Surface Velocity Profile of Air in the Proposed Designs.

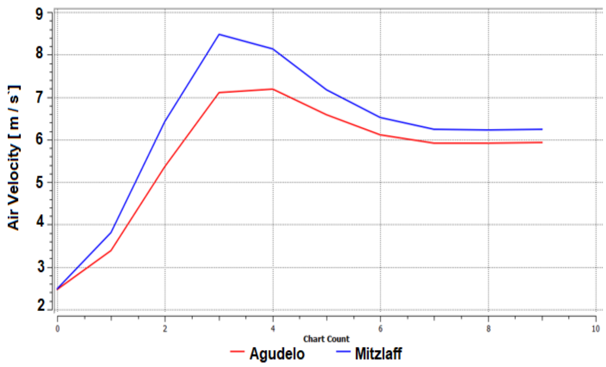


Fig. 8: Surface Velocity Profile in Air-Biogas Mixers According to the Longitudinal Axis of the Air Stream.

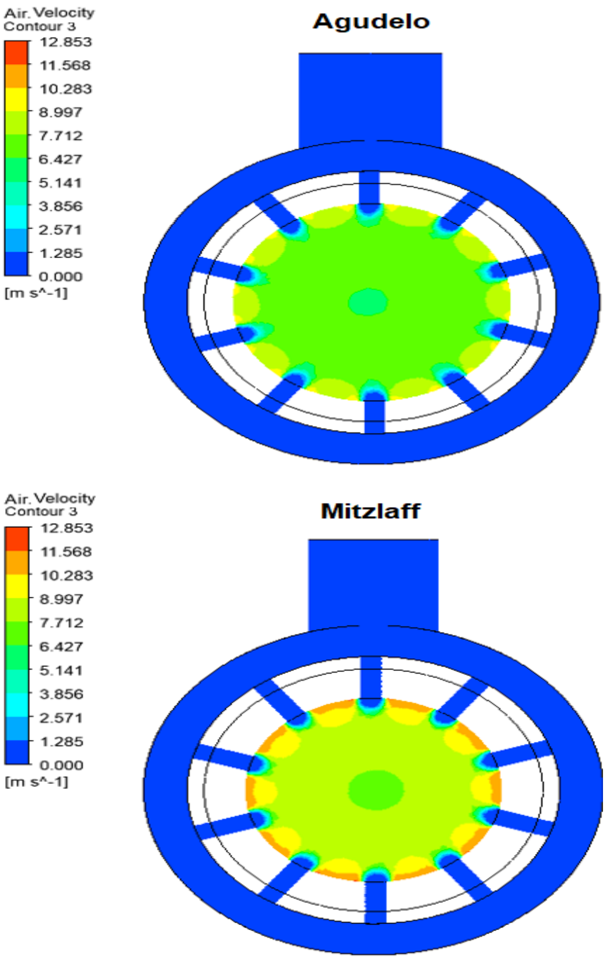


Fig. 9: View of Section in Throat and Contours of Surface Velocity of Air in Mixer.

4. Conclusion

This research used CFD to numerically solve the conservation equations that govern the mixing process of air and biogas in mixers sized for two analytical design methodologies, to compare them hydraulically and estimate which is more favorable for the process, taking into account turbulence models and auxiliary equations in the design. For this purpose, the design approaches of Agudelo-Mejia and Mitzlaff were taken for CAD modeling and then, the volume that would occupy the fluids in these devices was simulated under the same boundary conditions, with the objective of verifying the trends in speed and pressure in regions of interest of the mixers.

It was observed that in the Mitzlaff methodology there is a tendency to distribute the biogas in greater proportion over the divergence part of the mixer than that proposed by Agudelo. Likewise, the Mitzlaff design proposal tends to produce greater pressure drops in the throat section of the mixer, compared with the Agudelo methodology. Finally, it was found that the Agudelo-Mejia design approach tends to induce biogas in the throat of the device at pressures above 253Pa with respect to the Mitzlaff design strategy and it was detected that the gradients of surface velocity of the air in the case of Mitzlaff are of greater magnitude with respect to the radius of the throat than when compared with the methodology of Agudelo.

References

[1] J. Agudelo, R. Mejía. (2001) Desarrollo de un Modelo para el dimensionamiento de mezcladores aire-gas natural para motores, Revista Facultad Ingeniería de la Universidad de Antioquía.
[2] V. Mitzlaff. (2008) Engines for Biogas, Theory, Modification, Economy, Operation, Ed. Gate, 59-69.
[3] AC. Chandekar, BK. Debnath. (2018) Computational investigation of air-biogas mixing device for different biogas substitutions and engine load variations, Renewable Energy, 811-824. https://doi.org/10.1016/j.renene.2018.05.003.
[4] F. Bermejo, W. Orozco. (2010) Diseño de un Mezclador Aire-Biogás para un Motor Diesel Turboalimentado, Prospectiva 8, 37-43.
[5] W. Pulkrabek. (2004) Engineering Fundamentals of the Internal Combustion Engine, 2th edition, Editorial Pearson Prentice Hall. https://doi.org/10.1115/1.1669459.
[6] J. Aditya, K. Umesh. (2013) Numerical Validation of Producer Gas Carburetor, International Journal of Engineering and Science Research 3(10), 4757-4766.
[7] S. Biradar, Ebinezar, R. Reddy. (2013) Validation of Producer Gas Carburetor Using CFD, International Journal of Latest Research in Science and Technology 2 (6), 90-94.
[8] K. Peda, K. Somasundaram, U. Gokulraj, B. Ashok, S. Denis, C. Ramesh. (2015) A Comparative Study of Port Injected And Carbureted Type Lpg-Diesel Dual Fuel Engine Using CFD Analysis, Journal of Chemical and Pharmaceutical Sciences 9, 328-334.
[9] MR. Sanket, H. Shinde. (2016) Overview of Design Considerations for Biogas Operated Intake Device, International Journal of Research Publications In Engineering And Technology 2 (7), 23-28.
[10] F. Vidian, F. Edianto, Ismail. (2018) CFD simulation 3D premixed combustion of methane: influence of the excess air, International Journal of Engineering & Technology, 7 (4), 5399-5403.