Performance of a venturi scrubber in cleaning the gas produced by gasification using Volume of Fluid (VOF) model: Multi-phase flow simulation

Haouari Khadra1, Rahmani Kouider1, Benmessaoud Tahar2,3, Naas Toufik Tayeb1, Rouibah Abdelkader1, A. Al-Kassir4, J.P. Carrasco-Amador4, Chati Tounsi1

1Laboratoire de Recherche Modélisation Simulation et Optimisation des Systèmes Complexes Réels. E-mails: k_haouari@univ-djelfa.dz, kouderrah1@gmail.com, toufiknaas@gmail.com, rouibah_a@yahoo.fr, tounsi1978chati@gmail.com
2Faculty of Sciences and Technology, Ziane Achour University, 17000 Djelfa, Algeria
3Ingenium Research Group, Universidad Castilla-La Mancha, 13071 Ciudad Real, Spain; Corresponding author e-mail: t.benmessaoud@gmail.com
4School of Industrial Engineering, University of Extremadura, Spain E-mail: aawf@unex.es, jpcarrasco@unex.es

Abstract—Venturi system widely utilizes in industrial applications to clean the gas and reduces pollutions. Therefore, many studies have been conducted, especially in terms of pressure drop because it is one of the main parameters to determine its efficiency. But the phenomenon of mass transfer in the venturi scrubber did not get much attention. In this work, mass transfer was studied through a two-dimensional simulation of cleaning the gas produced by the gasification of biomass in Venturi scrubber with boundary conditions represented in air inlet velocities of 10, 15, and 20 m/s and water inlet mass flow of 0.02, 0.04 and 0.06 kg/s. Navier-Stokes equations were solved numerically and the mass transfer technique was treated by the Volume Of Fluid (VOF) model, using CFD software. The mass fraction, velocity and pressure contours, and profiles, were presented to analyze the obtained results. Also, the Probability Density Function (PDF) of mass transfer was investigated. The results show that the proposed venturi has the best mass transfer performance with a PDF that reaches 97.67 for velocity liquid of 20 m/s. In addition, the removal efficiency is higher with the decrease of liquid flow rates.


I. INTRODUCTION

Cleaning the gas resulting from industrial applications is very important. To achieve this objective venturi shape is one of the most device used because of its high efficiency in removing fine particles, its low cost, ease of maintenance, and its simple design. The venturi consists of three principal sections, convergence, diffuser, and the throat. It also contains orifices for the liquid introduction, which may be in the throat or before the convergence section. Venturi uses the high energy produced by accelerating the gas in the convergence section to break up the liquid into droplets in the throat. These droplets are responsible for cleaning the gas as they encapsulate and trap the particles [1][2].

Given the importance of venturi scrubbers, it has received a lot of interest from many mathematical and computational model have been developed a study to study the parameters affecting venturi scrubber. Lina [3] provided a study on the influence of venturi on the cleaning performance of elliptical filter cartridge. Sunuk et al [4] proposed a thermal cyclic test device with a Venturi injector designed to test a large number of samples rapidly. Kenneth [5] presented a paper based on Boll's model containing a set of design charts and equations to give general solutions for estimating pressure drop in Venturi devices. Kailash [2] investigated the performance evaluation of venturi aeration system. Breitenmoser et al [1] presented a model to predict the efficiency of droplet size distribution in a Venturi scrubber with liquid film injection. Viswanathan [6] developed an annular flow model to predict the pressure drop in venturi considers venturi geometry, throat gas velocity, liquid to gas.
ratio, and liquid film flow rate. Through the integral momentum equation, Azzopardi [7] developed a model involving the growth of the boundary layer in the diffuser for Venturi to predict the pressure drop. Pulley et al [8] extended the model Azzopardi to include predicting the efficiency of particle collection in venturi scrubber. Ananthanarayanan et al [9] tested a model bi-dimensional simplified depending on the experimental data to predict collection efficiency and liquid flux distribution in a venturi scrubber, the effect of throat gas velocity, liquid to gas ratio, aspect ratio, and nozzle diameter. Gonçalves et al [10] evaluated the models available to predict pressure drop in venturi by comparing the mathematical equations for these models with experimental data. Mohebbi developed a two-dimensional model based on the (Eulerian/Lagrangian) method to predict the efficiency of particle collection from the gas stream in the one-orifice venturi scrubber [11]. Sunand Azzopardi [12] extended the boundary layer model in the diffuser section to include a description of the boundary layer in three venturi sections to predict the pressure loss for Venturi type Pearce–Anthony at high pressure. Viswanathan proposed an improved algorithm based on a prediction of minimum pressure drop prediction to improve Venturi performance. It takes into account the design and operating parameters. This algorithm was applied cylindrical and rectangular venturi devices of the type of Pease-Anthony [13]. Nasseh et al [14] relied on the artificial neural networks approach to predict the pressure drop in the venturi. He applied the design of three independent artificial neural networks using three data sets of five different venturi. Based on models of Clavert and Young, Alexa et al [15] were improving graphical tools to estimate the overall collection efficiency of Venturi devices. Under the defined operating conditions and design, Kumar has developed a mathematical model to study the drop Reynolds number effect on small particle collection in venturi scrubber [16]. Artificial neural network design was used by Taheri, based on a genetic algorithm, where He used experimental data to create these artificial neural networks to predict the collection efficiency of venturi scrubber [17]. Nasseh used neural networks to estimate the pressure loss in venturi scrubber with an annular two-phase flow model, genetic algorithm, and artificial neural networks. He used experimental data for differently designed venturi devices, three injection systems, and different operating conditions for design these networks [18]. Shraiber et al [19] developed a mathematical model that includes gas turbulence to study its effects on the aggregation efficiency in venturi scrubber. To study the fluid mechanics of the two liquid and gas phases and their effect on the pressure drop in Venturi, Guerra et al [20] performed a three-dimensional simulation on a rectangular venturi used the VOF model and evaluated the influence of the number of the orifices on the distribution of the liquid in venturi. Ali et al [21] performed a 3D simulation on venturi scrubber based on Euler’s model by CFX and presented his work through three scientific papers. The first paper focused on pressure lower and studied the effect of the rate flow gas and the static pressure in the liquid inlet. The second paper includes the evaluation of the pressure in the throat and the study of fluid dynamics in the venturi. The third paper contained the effect of gas and liquid mass transfer on dust removal efficiency [22] [23]. Toledo-Melchore et al [24] performed a three-dimensional simulation of different gas flows on five venturi geometries, the difference between them in the angles of convergence and divergence. And the two-phase simulation of a single venturi geometry to study the behavior of gas flow and the effect of water flow on pressure drop. Luan et al [25] achieved a 3D numerical simulation of the flow of a gas inside a square venturi scrubber using ANSYS-fluent and compared contours of velocity, pressure, and the wall shear stress with circular venturi scrubber. Qamar studied the effect of the number and location of holes on dust collection efficiency in the Venturi through numerical simulations based on the Euler-Lagrange approach using ANSYS–CFX [26]. Manisha Bal [27] simulated three-dimensional fluid flow in venturi scrubber by CFD for studying the effects of throat gas velocity, liquid mass flow rate, and liquid to the gas ratio on the pressure loss. used in this simulation a Reynolds reorganization group (RNG), k-ε turbulence model, and the volume of the fluid model (VOF). To examine the effect of the liquid to the gas ratio on the collection efficiency of the potassium oxide particles in Venturi, Safdar et al [28] made a 3D simulation using CFX for different streams of gas and liquid. Few studies have been concerned with thermal and mass transfer in Venturi. As model Placek’s model for determining the particle collection efficiency of the venturi scrubber includes heat and mass transfer and analyzes the mechanisms of particle collection [29]. Taheri et al [30] includes a 3D simulation of absorption of SO2 based on a mathematical model. And a study of the mass transfer process assuming that the liquid was a mixture of droplets and film. Rahimi et al [31], [32] developed one-dimensional model to analyze the effect of heat and mass transfer on efficiency in venturi devices and operating parameters’ effect on the phenomenon of heat and mass transfer. He presented another model by developing the equations governing the transfer of heat and mass between two stages. For analysis on laminar forced convection heat and mass transfer in tube venturi with the wetted wall. Igo et al [33] [34] conducted numerical simulation on the rectangular venturi, where he focused in his study on the influences of the venturi effect, inlet Reynolds number, and venturi diameter ratio. and he also studied turbulent flow in venturi, by a two-dimensional simulation of air turbulent flow. This work aims to study cleaning the gas produced by the gasification of biomass in Venturi and mass transfer in Venturi scrubber. Many authors had studied the gasification process of biomass to produce a clean gas as combustible for energy production, such as Al-Kassir et al [35] The use of biomass residues as non-pollutant
fuels in combustion processes has an economical benefit and a social development of rural zones decreases the environmental impact by eliminating the remnants and decreasing the emissions of pollutants to the atmosphere as deduced by Awf Al-Kassir et al. [36]. They developed a study on drying and combustion of solid biomass residues and its combustion inside a boiler to produce clean combustible gas. - Due to the difficulty of cleaning the gas resulting from industrial applications. Venturi shape is one of the most devices used because of its high efficiency in removing fine particles, its low cost, ease of maintenance, and its simple design. - Therefore, our contribution aims to improve high mass transfer and cleaning performances for the gasification of biomass fluid inside a new venturi scrubber.

II. GOMETRY DETAILS AND MODELING

II.1. Geometry description

Figure 1 presents the 2D geometry utilized in this work. Where venturi geometry consisting of a convergent and a diffuser section with a diameter of 0.075 m. A throat section with a diameter of 0.027 m. The length of the convergent section, diffuser, and throat are 0.13, 0.34, 0.165 m, respectively (As show in table I).

The boundary conditions are to impose a velocity of the gas on the gas inlet with a volumetric fraction equal to zero. Velocity of the liquid at the liquid inlet with a volumetric fraction equals one, and static pressure at outlet equal zero.

Apply no-slip velocity condition at wall surfaces.

II.2. Properties of gas

Table II includes the properties of the gas produced by the gasification of biomass used in this study [37]:

| Density (kg/m³) | 1.03 |  
| Molar specific heat capacity (kJ/(mol·K)) | 22.05 |  
| Molar mass (kg/mol) | 25.14 |  
| Viscosity (kg/(m·s)) | 1.57263x10⁻⁵ |  
| Thermal conductivity (W/(m·K)) | 0.0344 |  
| Diffusion coefficient (m²/s) | 0.209x10⁻⁶ |  

II.3. Governing equations

The continuity, momentum, diffusion, and species transfer equations that express the governing equations for Newtonian, incompressible, and immiscible flows, are defined as follows [1] [2]:

\[ \nabla \cdot \mathbf{u} = 0 \]  

\[ \frac{\partial \rho \mathbf{u}}{\partial t} + \nabla \cdot (\rho \mathbf{u} \otimes \mathbf{u}) = -\nabla p + \mu \nabla^2 \mathbf{u} + \rho g \]  

\[ \frac{\partial c}{\partial t} + \nabla \cdot (c \mathbf{u}) = D \nabla^2 c \]

\( \rho \) is the total density, \( t \) the time, \( p \) the pressure, \( g \) the gravitational acceleration, \( D \) is the species mass diffusivity, \( c \) is molar concentration.

The pressure drop must be taken into account with the removal efficiency and the flow rate when evaluating the overall performances of the cleaning venturi reliability. For this, it’s important to present the Removal Energy Cost “REC” which is expressed by:
The removal efficiency is determined from the following equation:

$$E_R = \frac{C_m - C_0}{C_m}$$  \hspace{1cm} (5)

Where $Q$ is the flow rate (m$^3$/s), $\Delta P$ is the pressure drop along the venturi and $E_R$ is the removal efficiency of the proposed venturi. $C$ is the mass fraction of water.

II. MESHING VALID

The grid independency has an essential role in simulating CFD to obtain approximate results. So, choosing the best network is very important. Four different meshes have been identified for simulation. The nodes number in these models is as follows: 13557, 31345, 81938 and 144696. The simulation was in these models with a velocity gas is 20m / s, and a mass flow is 0.02kg / s. The velocity variations evaluate for increasing nodes as shown in Figure 2. As we note, there is no significant difference between the different mesh results. As a result, the mesh with 81938 nodes adopted as a suitable mesh for investigation.

Numerical simulations of air and water flow in Venturi scrubber performed using CFD code. The multiphase flow is processed using the VOF model. The solutions in this simulation are to use the explicit schema for time estimation, application of the PISO algorithm for pressure, velocity coupling and PRESTO pressure diagram. The second-order upwind scheme uses for the discretization momentum equations.

III. RESULTS AND DISCUSSIONS

Pressure, velocity, and mass fraction are the most important parameters in evaluating the performance of a venturi scrubber. In this study, a numerical simulation of Venturi cleaning flow using CFD code. The simulation results depend on boundary conditions represented in gas velocities of 10, 15, and 20 m / s and mass flow liquid 0.02, 0.04, and 0.06kg / s.

Figure 3 shows the velocity evolution in the venturi as a function of the change in inlet gas velocity and constant inlet water mass flow equal to 0.06kg/s. The three curves have the same behavior. In the range from 0.7 to 0.5, we notice an increase in the velocity until it arrives at the water inlet. Then it decreases because there is a contraction in the airway at the water inlet. The velocity increases in the range converging section from 0.7 to 0.6 m/s due to the section diameter change according to the principle of mass continuity. Also, it continues to increase until it arrives at its maximum value in the field from 0.5 to 0.35 m / s, due to the friction between the maximum fluid velocity and the wall in the throat section. We notice the velocity decrease in the diffuser range from 0.35 to 0 m /s because the change in the section diameter. Figures 4 and 5 represent the contours and the graphs of the velocity of flow multiphase in venture scrubber at gas velocities inlet 10, 15, and 20 m / s and at inlet water mass flow 0.02, 0.04, and 0.06 kg/s. We observe that the increase in inlet velocity influences the velocity development in venture scrubbers. The velocity
arrives at 30.81, 32.67, and 34.36 m/s at the gas inlet velocity of 10 m/s at the inlet water mass flow 0.02, 0.04, and 0.06 kg/s, respectively. It arrives at 45.08, 47.04, and 48.84 m/s at the gas inlet velocity of 15 m/s at the inlet water mass flow 0.02, 0.04, and 0.06 kg/s, respectively. It arrives at 59.33, 61.3, and 63.22 m/s at the gas inlet velocity of 20 m/s at inlet water mass flow 0.02, 0.04, and 0.06 kg/s, respectively.

Figure 4. Velocity profiles as a function of different inlet water velocities at the middle vertical line of the venturi

Figure 6 indicates the contour of static pressure in the venturi. The static pressure is high at the beginning of the venturi. It begins to decrease in the convergent section due to the conversion of static pressure into kinetic energy. There is a sharp drop of static pressure due to friction in the throat. Part of the pressure is recovered in the diffuser due to the conversion of kinetic energy into pressure.
Figure 6. Contours of static pressure

Figure 7. Comparison of static pressure with various water velocities.
Figure 8. Comparison of dynamic pressure with various water velocities.

Figure 9. Contours of dynamic pressure
The results of the comparison of static pressure with mass flow difference for inlet water mass flow and static gas velocity inlet are represented in figure 7 (a) where we note, at the gas velocity inlet of 10 m / s and the inlet water mass flow 0.02, 0.04, and 0.06 kg/s, static pressure approximate values at the beginning of the Venturi were 93.29, 109.06, and 117.96 (Pa), respectively. The minimum value of static pressure at the throat is approximately -606.73 Pa for the three mass flows at the water inlet. For air velocity inlet of 15 m / s and the inlet water mass flow 0.02, 0.04, and 0.06 kg/s, static pressure approximate values at the beginning of the Venturi were 298.69, 335.22, and 369.14 (Pa), respectively. The minimum value of static pressure at the throat is approximately -1239.73 Pa for the three velocities of water at the inlet.

The mass fraction of the water contour represents in Figure 9. In all cases, the mass fraction is high at the water inlet, especially with the mass flow water inlet 0.06 kg/s then concentrates on the wall under the influence of the high velocity of the gas.

Figure 11 illustrates the mass transfer for different inlet water velocity cases at the throat of the venturi. It has been observed that the mass fraction is more enhance with increasing in water velocity in the throat section of venturi scrubber for all cases of inlet gas flows. The mass fraction of gas downstream of orifices gradually reduces because of scrubbing water solution entering from orifices, see figure 12. This increases the importance of the proposed venturi efficiency.

Figure 12 expresses the probability density function for the volume fraction of water with the velocity difference of water mass flow inlet and a constant air velocity inlet in a venturi. We see that the large percentages of PDF were with a volume fraction less than 0.1 for all cases. This result indicates that the mass transfer in the venturi scrubber is weak.
Figure 11. Mass fraction at X-position in throat of venturi.

Figure 12. PDF of mass fraction with the different mass flow of water.
Figure 13 shows the difference in removal efficiency at different mass flow rates. It has been observed that the removal efficiency of the venturi scrubber increases with a decrease in the liquid flow rate and an increase in the gas velocity.

Figure 14 illustrates the evolution of the cost of removal energy of the venturi system for various cases of inlet mass flow rates. As can be seen, the REC increases with increasing in the inlet water flow rate and increasing in gas velocity.

IV. CONCLUSION

was applied numerically for a gas generated from biomass gasifiers. Through the results, we can conclude that:

- The liquid flow affects the velocity as the velocity increases with the increase of the liquid flow with the same velocity of the air.
- The dynamic has the same behavior velocity, Where decreases with the increase of gas velocity and water velocity at inlets.
- The mass fraction of the water concentrates on the wall under the influence of the high velocity of the gas.
- The effect of mass transfer in the venturi scrubber is weak.
- The removal efficiency of the venturi scrubber increases with a decrease in the liquid flow rate and an increase in the gas velocity.
- The REC increases with increasing in the inlet water flow rate and increasing in gas velocity.

We propose these suggestions as future work:

- Investigate the mass transfer through a three-dimensional simulation of cleaning the gas produced by the gasification of biomass.
- Create a new Venturi scrubber with different boundary conditions to enhance more the energy efficiency.
- Study the influence of venturi on the cleaning performance of nano systems.

REFERENCES

Pressure drop and liquid distribution in a venturi scrubber: Experimental data


Numerical simulation of turbulent forced convection in a venturi channel with fully developed flow at the inlet Numerical simulation of turbulent forced convection in a venturi channel with fully developed flow at the inlet, no. January 2014, 2015.


