ENHANCED PERFORMANCE OF KNOCK OUT DRUMS BY FEED DISTRIBUTOR DESIGNED WITH CFD ASSISTANCE

E.B. QUADRO† and L.L.N. GUARIEIRO ‡

† Braskem, Rua Eteno, 1561, Pólo Petroquímico de Camaçari, 42810-000, Camaçari, BA, Brasil.
emerentino.quadro@braskem.com
‡ Faculdade de Tecnologia SENAI CIMATEC, 41650-010, Salvador, BA, Brasil.
lilian.guarieiro@fieb.org.br

***Abstract*— The big difference between maximum and minimum up flow velocity in a fuel gas Knock Out Drum (without feed distributor) explains the poor performance of this industrial equipment. The up flow fluid velocity profile was determined with CFD assistance for two models of feed distributors. The highest up flow velocity was reduced from 5.1 m/s to 1.9 m/s for the proposed feed distributor. Calculations have showed the capacity of the feed distributor to reduce the maximum droplet diameter carried by the fuel gas from 1,532 µm to 675 µm. This device was built and installed in an industrial equipment. The Residence Time Distribution (RTD) was measured by analyzing the response to a tracer pulse injection. The RTD data has confirmed the improvement of flow uniformity inside the Knock Out Drum. Besides, the maintenance team feedback has confirmed the reduction of 58% of burner's failure rate relative to the previous value.**

***Keywords*— Computational Fluid Dynamics; feed distributor; Knock Out Drum; liquid entrainment; Residence Time Distribution.**

I. INTRODUCTION

Many refineries and petrochemicals plants produce the fuel gas consumed in their own process. This fuel gas is composed basically of methane and hydrogen. The separation of fuel gas from its impurities, including liquid droplets, is a key issue for the modern and sensitive low NOx burners (Platvoet and Baukal, 2013; Dragomir *et al.,* 2010).

There are many types of liquid impurities present in the fuel gas, causing trouble for the burners (Sazhin *et al.*, 2006), the main one is the heavy oil formed in the catalytic reactors and adsorbent beds, known as green oil, and removed from these beds during the regeneration procedure. Methane is usually used for stripping these beds (the first step of the regeneration procedure), before it is sent to the fuel gas pool. The most common approach to remove this oil from the fuel gas sent to the burners is to feed the fuel gas into a Knock Out Drum (Jekel *et al.*, 2001).

However, after a plant revamp project (plant capacity increasing) the capacity of several equipments is pushed to the limit. One of these equipments is the fuel gas Knock Out Drum that may become too small to ensure fuel gas proper quality. In order to overcome this limitation, many kinds of internal devices could be used, including mist eliminators and inlet flow distributors (Al-Dughaither *et al.*, 2010). Eventually, mist eliminators should be avoided due to the risk of oil accumulation in this device, resulting in a too high pressure drop. An excessive pressure drop in this part of the process implies an undesired plant shut down. In this case, a well-designed feed flow distributor could save the investment to replace the equipment for a bigger one.

In order to solve an industrial issue (poor performance of fuel gas Knock Out Drum), the flow distribution was studied for two designs of flow distributors by carrying out virtual experiments. It has been recognized that flow distribution uniformity is associated to a proper Knock Out Drum performance (Bagul *et al.*, 2013).

Virtual experiments, when applied to problems involving fluid motion, could be carried by using Computational Fluid Dynamics, or CFD, a set of numerical resources used to describe the momentum, energy and mass transfer, associated with fluid flow and its interactions with the physical geometries (Vedovoto *et al.*, 2015). CFD is currently an important interdisciplinary tool (Mavriplis, 2012), in particular, it has been applied to achieve a better performance in many industrial equipment where a high uniformity of flow distribution is required (Oliveira *et al.*, 2012). CFD can reduce drastically the number of necessary experiments to develop projects and consequently reduces the cost of implementing them (Patankar, 1980; Ranade *et al.*, 2011).

The Navier-Stokes equations are solved simultaneously in an iterative process to achieve a final solution. However other equations (kinetic models, vaporization, condensation, adsorption turbulence etc.) can be associated until a suitable description of the physical process is produced (Maliska, 2010). CFD techniques were developed in the most general way possible, to be used in almost any industrial application to allow the user to visualize the handling of equipment in a way that cannot be matched by real world measurements.

This investigation uses RTD (Residence Time Distribution) to characterize the flow within the vessel. RTD is an useful resource to evaluate how uniform is the velocity distribution inside the equipment (Naumann, 2008; Martin, 2000; Froment and Bishoff, 1990).

The main objective of this study is to present a methodology, based on CFD, to evaluate the influence of the feed flow distributor on the maximum droplet diameter carried by fuel gas in this Knock Out Drum. This methodology was applied in a troubleshooting related to an under designed fuel gas Knock Out Drum, currently in operation in a petrochemical plant. This issue has caused damage in the fuel gas burners, resulting in high maintenance cost and high rate of burners replacement.

II. METHODOLOGY

A. Geometrical configuration and specifications of the drum considered

A simplified drawing of a typical Knock Out Drum is shown in Fig. 1. However, the fuel gas Knock Out Drum analyzed in this study does not include a mist eliminator. Its absence is due to the risk of blocking it with oil accumulation.

The pressure drop in the Knock Out Drum, during the entire plant run length, should be kept as low as possible. This device must work continuously for five or six years without any kind of maintenance, in order to match with the industrial plant run length. The internal design of a fuel gas Knock Out Drum should be kept as simple as possible to avoid potential blockages due to internal failure or fouling accumulation. The feed flow distributor should not retain oil, or eventual solids, fed together with the fuel gas.

In the system investigated, liquid concentration in the fuel gas is usually lower than 0.05% weight. This information has been obtained experimentally by measuring the quantity of oil drained from the fuel gas Knock Out Drum compared with the quantity of fuel gas fed to it in a same period.

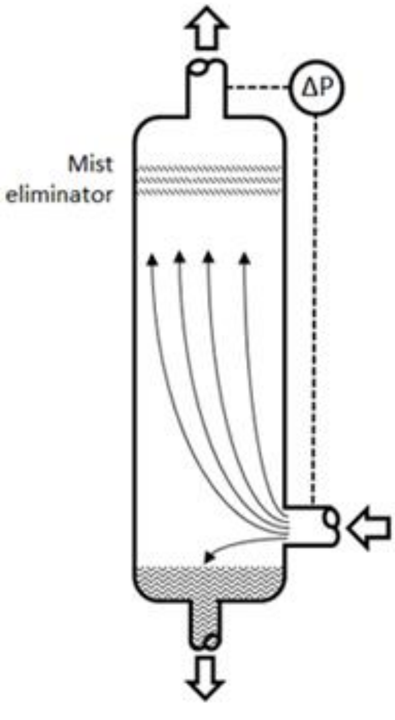

Fig. 1. Simplified representation of a typical Knock Out Drum. (Source: author).

Table 1. Oil and gas characteristics, Knock Out Drum operation pressure and temperature and vessel diameter.

Knock Out Drum feed temperature		27.0°C
Knock Out Drum feed pressure		3.50 kgf/cm^{2}g
Fuel gas viscosity at feed process conditions		0.0114 cP
Fuel gas mass density at feed process conditions		2.184 kg/m^3
Liquid mass density		1,012 kg/m^3
Knock Out Drum internal diameter		1,980 mm
Fuel gas composition	Methane	95% weight
	Hydrogen	5% weight

Initially, green oil characteristics were determined based on several samples collected from the drain located at a low point of the fuel gas header, downstream the fuel gas Knock Out Drum (where it is not expected, neither desired, finding oil accumulated) . This header is responsible for sending fuel gas to the burners. In fact, it is not desirable to find oil in this header. It happens due to the poor knock out of the oil present in the fuel gas (this Knock Out Drum is too small for the required service).

The basis of calculation for the fuel gas composition was determined by an on line chromatograph analyzer, with FID detector. Fuel gas properties (mass density and viscosity) were estimated by Hysys Process Simulator (property package: Peng-Robinson, an appropriate VLE equation to low pressure system formed by non-polar hydrocarbons) (Ramdharee *et al.*, 2013). Fuel gas flow rate assumed in the calculation was 30,000 kg/h. Oil and gas characteristics, Knock Out Drum operation, pressure and temperature, besides vessel diameter, are shown in Table 1.

After the simulation of the Knock Out Drum "as is", without any feed distributor, it was simulated again with the selected models of feed distributor under the same process conditions. Two models of feed distributor were evaluated, as shown in Fig. 2.

The Model 1 is a curved plate, parallel to the vessel wall (280 mm from it), located inside the vessel, just in front of the inlet nozzle. This curved plate has 42 holes with 10 mm diameter. The Model 2 is formed by 10 vertical vanes, located inside the vessel, just in front of the inlet nozzle. The angles of these vanes favor to reduce the momentum and to distribute the flow uniformly.

The free areas for both feed distributor models are big enough to minimize the possibility of fouling accumulation during the plant run length, typically between 4 and 6 years.

B. CFD model and numerical approach

In fact, the stream fed to the Knock Out Drum has two phases (gas and oil). ANSYS Fluent includes several multiphase models, in Lagrangian and Eulerian approaches (ANSYS, 2011). However, this CFD simulation will assume a single phase stream, without considering the presence of oil droplets in the fuel gas. This simplification is justified by the low oil concentration in the fuel gas, not big enough to modify the velocity field inside the Knock Out Drum. The assumption of single phase to determine numerically the flowing characteristics is reasonable, once it is not expected these droplets could change the flowing characteristics or velocities field.

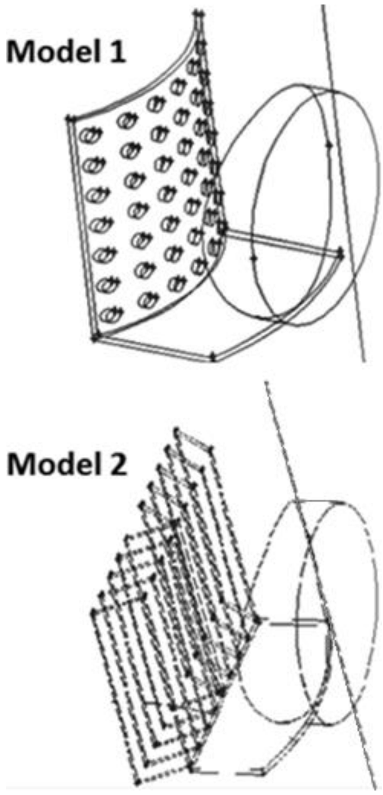

Fig. 2 - Feed distributor models evaluated.

Incompressible Navier-Stokes equations (represented in the Eq.1 in a general form, without energy balance) were used to model the gas flow inside the Knock Out Drum at steady state condition. The CFD calculations were performed in a Finite Volume approach by the commercial software ANSYS Fluent in a steady state mode, 3D, single phase and double precision. It was employed a pressure-based Navier-Stokes solution algorithm. The turbulence model applied was the k-ε and the density was calculated by the Peng–Robinson VLE model. The calculation of energy in the model was disabled, once there is no reaction and the system is adiabatic.

The non-structured 3D grid was formed by 3 million tetrahedra, built by software ICEM CFD Fluent. Usually a non-structured grid is recommended for complex geometry (Fell, 2009). The criteria adopted to ensure minimum grid quality are listed in Table 2.

$$\rho\frac{\partial u}{\partial t} - \nabla\cdot\eta\left(\nabla u + (\nabla u)^{T}\right) + \rho(u\cdot\nabla)u + \nabla p = 0 \qquad (1)$$

$$\nabla\cdot u = 0$$

where η denotes the dynamic viscosity, u the velocity vector, ρ the density of the fluid, and p is the pressure. Once the simulation was run in a steady state mode, $\partial u/ \partial t$=0.

It was performed a grid sensitivity analysis, and confirmed that a higher resolution discretization does not affect the results. To confirm the grid-independent velocities profiles under specified operation conditions, it was verified the simulations results varied less than 0.1%, after the number of cells has been increased from 3 to 4 million tetraedra.

Table 2. Grid quality criteria. (Zikanov, 2010)

Parameter	Reference
Skewness:	<0,95
Aspect ratio of tetrahedral cells:	<5
Minimum angle:	>18
Orthogonal quality:	>0,05

The maximum scaled residual (calculated by Fluent) accepted in simulation for mass imbalance (continuity), momentum and turbulence parameters (k and ε) were 1.10^{-4}. However, this is not enough to guarantee the convergence has been reached. Once achieved this residual, it was run more 10^{3} iterations to ensure the variation of the velocity in a same point is lower than 0.1%. Inlet process conditions were set, from where the simulation started (the simulation has started from inlet).

C. Determination of maximum droplet carry over

Once defined the droplet mass density and assumed the perfect spherical shape for this droplet, the maximum droplet diameter carried over is the function of the vertical fluid velocity. The uniformity of the velocity is analyzed at 1,500 mm above the inlet nozzle center line. Above this point it is expected to start the outlet vortex.

The calculation strategy was (i) to determine the maximum vertical velocity using CFD techniques at 1,500 mm above the inlet nozzle and (ii) to calculate the maximum droplet diameter for which its own weight (w) is equal to drag force (FD) to the maximum vertical fluid velocity, theoretical scenery when the droplet reaches a static equilibrium.

The drag force exerted by the fluid on a droplet, of whatever format, can be calculated by the Eq. 2 (Perry and Green, 1997), while the droplet weight is given by the Eq. 3.

$$F_D = \frac{C_D A_p \rho\, u^2}{2} \qquad (2)$$

where F_D is the drag force exerted by the fluid flowing on the droplet, C_D is the drag force coefficient, A_p is the droplet projected area on the plane orthogonal to the flow direction, ρ is the fuel gas mass density and u is the relative velocity between the droplet and the flow, considering only the vertical component for both, droplet and fluid velocities.

$$p = g m_p \qquad (3)$$

where p is the droplet weight, g is the gravitational acceleration (9.81 m/s^2) and m_p is the droplet mass.

The limit condition to knock out the droplet is that its weight should be, at least, equal to the drag force applied by the fluid on the droplet, as described in Eq. 4. This equation describes the forces equilibrium acting on the droplet when the weight of the droplet has to be balanced by the upward drag force.

$$F_D = p\,. \qquad (4)$$

Here is introduced a simplification. It has been assumed that droplets are perfectly spherical. Based on this assumption and replacing the Eq. 2 and Eq. 3 in the Eq. 3, the droplet entrainment phenomenon could be described mathematically by the Eq. 5.

$$C_D \rho\, u^2 \pi \frac{D^2}{8} = \rho_d \left(\frac{4}{3}\right)\pi\left(\frac{D}{2}\right)^3 g. \cdot \tag{5}$$

where D is the droplet diameter and ρ_d is the droplet mass density. C_D is a function of $Re_p=u\rho D/\mu$. In the intermediate regime ($0.1 < Re_p < 1{,}000$), the drag force coefficient may be estimated within 6 percent of accuracy by the Eq. 6 (Perry and Green, 1997).

$$C_D = \left(24\big/Re_p\right)\left[1+0.14Re_p^{0.7}\right]. \tag{6}$$

So, the Eq. 5 can be rewritten, resulting in the Eq. 7.

$$\left(18\big/Re_p\right)\left(1+0.14Re_p^{0.7}\right)\rho u^2 = \rho_d D g. \cdot \tag{7}$$

Once all the terms of the Eq. 7 were determined, except the droplet diameter (D), we are able to solve this equation by using the maximum vertical velocity identified by CFD simulation at the referred height. The Newton's method was used to solve this equation.

So, D is the maximum droplet diameter carried over by the fluid in the Knock Out Drum. The maximum allowed droplet diameter carried over depends on the burner design. In this case, based on burner manufacturer recommendation, the fuel gas Knock Out Drum should retain droplet with diameter above 50 μm.

D. RTD measurements

The performances of two different designs of feed flow distributor were quantitatively compared with the "as is" situation (no feed flow distributor).

Once the feed distributor is able to achieve a more uniform flow distribution, it is expected this device provides a reduction of the maximum droplet diameter carried by fuel gas. It was not possible to measure directly the liquid droplet diameter distribution inside the drum and neither were studied the factors that affect this droplet diameter spectrum. It just has been assumed that the feed distributor doesn't affect the droplet diameter spectrum.

The effectiveness of the Knock Out Drum modifications was quantified by Residence Time Distribution (RTD) experiments. The methodology consisted in introducing almost instantly a very small volume of a concentrated tracer at the inlet (a pulse). Although an infinitely short injection cannot be produced, it can be made a much smaller one than the mean residence time in the drum (Choi *et al.*, 2004; Lopes *et al.*, 2002).

The RTD was determined experimentally by injecting a non-reactive chemical tracer (a bullet of carbon monoxide, pressurized at 20 kgf/cm^{2}g) in the inlet of the Knock Out Drum. The tracer concentration was measured in the equipment outlet. The small quantity of tracer is not enough to modify the physical characteristics of the fluid (density, viscosity) neither the fluid dynamic characteristics.

If a mass of tracer is introduced into the equipment feed, the resulting curve of outlet tracer concentration, $C(t)$, can be transformed into a dimensionless residence time distribution curve, $E(t)$, by the Eq. 8.

$$E(t) = \frac{C(t)}{\int_0^\infty C(t)dt} \cdot \tag{8}$$

The function $E(t)$ has the property described by the Eq. 9.

$$\int_0^\infty E(t)dt = 1. \cdot \tag{9}$$

The RTD variance, or square of the standard deviation of the RTD, is calculated using the Eq. 10 (Choi *et al.*, 2004).

$$\sigma^2 = \int_0^\infty (t-t_m)^2 E(t)dt\,, \tag{10}$$

where t_m is the mean residence time, which is calculated by integrating the Eq. 11.

$$t_m = \int_0^\infty tE(t)dt \cdot \tag{11}$$

Qualitatively, the impact resulted from the modifications implemented in the fuel gas Knock Out Drum was verified during the furnace burners routine inspections. It has been observed that the high quantity of oil present in the fuel gas fed to the burners is responsible for the high frequency of burner malfunctions, resulting in a high rate of burner's replacement.

III. RESULTS AND DISCUSSIONS

The up flow fluid velocity contours in a cross section cut of a 3D simulation at 1,500 mm above the feed nozzle is shown in Fig. 3 for the base case (without feed distributor), with feed distributor Model 1 and with feed distributor Model 2. The big difference between maximum and minimum velocity for the base case explains its poor performance. The Model 2 has presented a more uniform velocity distribution, compared to Model 1 and base case. Above this point, the outlet vortex will increase its velocity dispersion.

Table 3 summarizes the calculation results. The feed distributor Model 1 is able to reduce the maximum droplet diameter carried over from 1,532 μm (base case) to 1,133 μm. However, feed distributor Model 2, more efficient than Model 1, reduces the maximum droplet diameter carried over to 675 μm. The better performance of Model 2 is due to the reduction of maximum vertical velocity, from 5.1 m/s in the base case to 2.6 m/s. The Reynolds number was calculated based on maximum up flow velocity in the reference cut (1,500 mm above the feed nozzle).

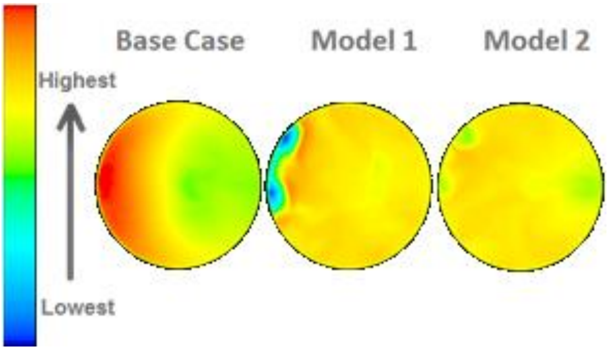

Fig. 3 - Vertical fluid velocity contours (1,500 mm above feed nozzle).

Table 3. Calculation Summary.

Parameter	No feed distributor Base case	Feed distributor Model 1	Feed distributor Model 2
Maximum velocity (m/s)	5.1	4.0	2.6
Reynolds number	1,307	758	293
Carry over coefficient (Cd)	0.409	0.491	0.693
Droplet drag force (µN)	1.3915	0.2557	0.0097
Maximum droplet diameter carried over (µm)	1,532	1,133	675

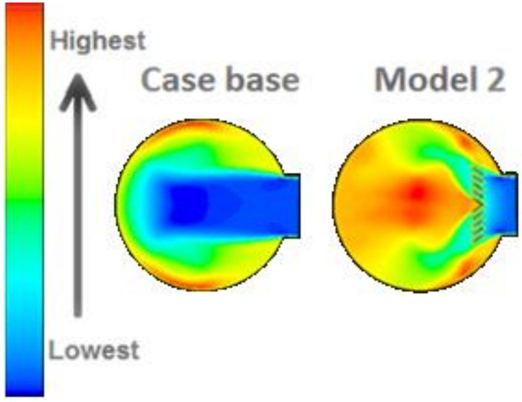

Fig. 4 - Horizontal fluid velocity contours (feed nozzle center line height).

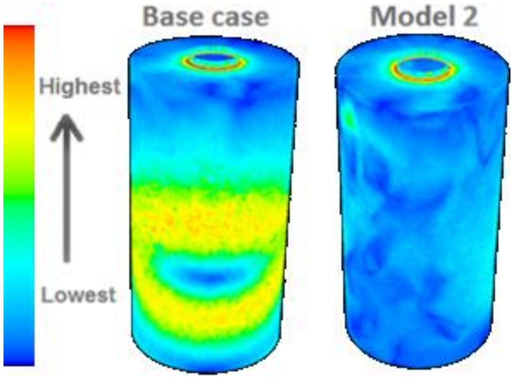

Fig. 5 - Wall shear stress at the opposite side of feed nozzle.

The numerical simulation predicted the knock out pressure drop. For the base case, comparing inlet and outlet, the calculated pressure drop is 22.7 mmH$_2$O. For Model 1, the estimated pressure drop is 26.6 mmH$_2$O and for Model 2, this value is 25.6 mmH$_2$O. In fact, the feed distributor causes such a small increment of pressure drop that is insignificant.

As shown in Fig. 4, the high fluid velocity through the feed distributor in Model 2 (around 11 m/s) results in two main concerns: Vibration and erosion. So, it is necessary to design proper supports in order to minimize vanes vibration and to use a hard material, once it is an erosive service.

The Fig. 5 shows the influence of the feed distributor (Model 2) on the wall shear stress, compared to the base case. This is an important issue when solids particles or droplets are present in the fuel gas fed to the equipment at high velocity, as it could results in wall erosion after years of continuous operation. In according to the CFD simulation, the maximum wall shear stress in the base case (0.0318 Pa, in the opposite side of the inlet nozzle) has reduced to 0.0172 Pa at the same process conditions when the feed distributor has been installed.

The feed distributor Model 2 was built and installed in an industrial fuel gas Knock Out Drum. Model 2 geometry details are represented in Fig. 6. The flowing uniformity could be verified by the experimental Residence Time Distribution.

The RTD variance (described by Eq. 10) for the base case, calculated from de response to a pulse injection, was 0.93 s^2. After the feed distributor installation, under the same process conditions, this RTD variance was reduced to 0.47 s^2. These results confirm an improved flow distribution inside the drum, consistent with the CFD simulation.

The proposed and installed feed distributor to the fuel gas Knock Out Drum has resulted in a significant oil entrainment reduction (based on the oil draining frequency from the fuel gas main header to the burners), but not enough to match the usual burner manufacturer recommendation (maximum droplet diameter: 50 µm), for what would be required a maximum vertical fluid velocity inside the drum lower than 0.80 m/s.

Additionally, based on the maintenance team feedback, the installation of the fuel gas Knock Out Drum feed distributor resulted in a significant reduction of fouling accumulation in the burners. Thereby, the MTBF (Medium Time Between Failures) for the burners, an important and usual maintenance index, has increased 72% (corresponding to 58% reduction of the burners failure rate in comparison to the previous value), resulting in a positive impacting on maintenance costs and equipment availability for operation. This information confirms the prognostic of the CFD simulation.

Due its simplicity, low cost and high mechanical robustness, the proposed feed distributor (Model 2) has advantage over other popular type of feed distributor, the schoepentoeter, a vane-type distributor developed by Shell (Kharoua *et al.*, 2013).

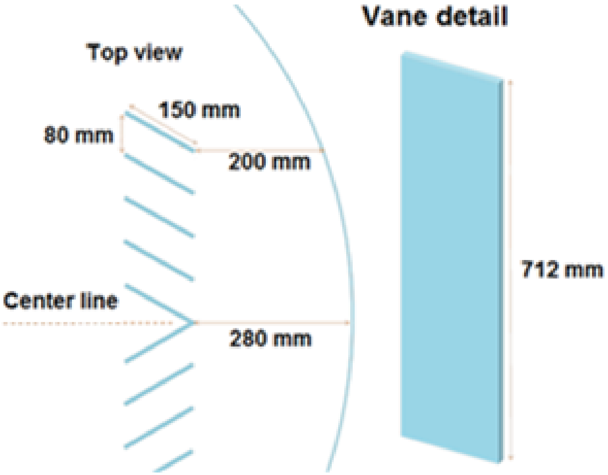

Fig. 6 – Details of the Model 2 feed distributor geometry.

IV. CONCLUSIONS

This study presents a methodology to evaluate the influence of the feed flow distributor on the maximum droplet diameter carried by the gas in an industrial Knock Out Drum, using CFD.

In order to illustrate its application, it has been evaluated the performance of two geometrical designs of feed distributors against a base case geometry of a fuel gas Knock Out Drum using single phase CFD simulation from the Fluent commercial package. Droplet equilibrium in a gas flow was used to determine the cut off size Experiments on residence time were implemented to substantiate the CFD findings.

The CFD simulation of the base case (without feed distributor) indicated that a droplet could be carried over, unless its diameter would be greater than 1532 µm. This result is consistent with the high burner's failure rate reported by the maintenance team.

Two different models of feed distributors were evaluated. The most efficient one was a simple and robust multi-vane type distributor. The CFD simulation has indicated this device could reduce the maximum droplet diameter carried by the fuel gas from 1532 to 675 µm, with a negligible additional pressure drop.

This feed distributor was installed in the industrial equipment. The influence of the feed distributor on the RTD, showed a significant improvement on velocity dispersion inside the Knock Out Drum, in according to the CFD prognostic. As consequence, the maintenance team feedback has confirmed the reduction of burner failure rate to 58% of the previous value, with significant impact on maintenance costs and equipment availability for operation.

ACKNOWLEDGEMENTS

The authors are grateful to Braskem, to the program Recursos Humanos of ANP/MCTI (PRH55), to the Faculty Senai Cimatec and to the FAPESB for their support to this investigation.

REFERENCES

Al-Dughaither, A.S., A.A. Ibrahim, W.A. Al-Masry. "Investigating droplet separation efficiency in wire-mesh mist eliminators in bubble column," *Journal of Saudi Chemical Society*, **14**, 331-339 (2010).

ANSYS, *Fluent User Guide and Fluent Theory Guide*, ANSYS Inc., version 14.0 (2011).

Bagul, R.K., D.S. Pilkhwal, P.K. Vijayan and J.B. Joshi, "Entrainment phenomenon in gas–liquid two-phase flow: A review," *Sadhana*, **38**, 1173–1217 (2013).

Choi, B.S., B. Wan, S. Philyaw, K. Dhanasekharan and T.A. Ring, "Residence Time Distributions in a Stirred Tank: Comparison of CFD Predictions with Experiment," *Ind. Eng. Chem. Res*. **43**, 6548–6556 (2004).

Dragomir, R., F. Drnevich, J. Morrow, V. Papavassiliou, G. Panuccio and R. Watwe, "Technologies for Enhancing Refinery Gas Value," *AIChE Spring Meeting*, San Antonio, TX, USA (2010).

Fell, B.J., *Structured Mesh Optimisation and CFD Simulation of the Fully Appended DARPA Suboff Model*, Australian Maritime College (2009).

Froment, G.F. and K.B. Bishoff, *Chemical Reactor Analysis and Design*, John Wiley (1990).

Jekel, T., D. Reindl and J.M. Fisher, "Gravity Separator Fundamentals and Design," *IIAR 2001*, *Ammonia Refrigeration Convention and Exhibition*, Long Beach (2001).

Kharoua, N., L. Khezzar and H. Saadawi, "CFD Modelling of a Horizontal Three-Phase Separator: A Population Balance Approach," *American Journal of Fluid Dynamics*, **3**, 101-118 (2013).

Lopes, J.M., F. Lemos, C. Pinheiro, F. Ramoã A. Ribeiro, F.D. Magalhães, A. Mendes and C. Costa, "Teaching Residence Time Distributions in the Laboratory," *Int. J. Engng*., **18**, 674–681 (2002).

Maliska, C.R., *Transferência de Calor e Mecânica dos Fluidos Computacional*, 2° ed. (2010).

Martin, A.D. "Interpretation of residence time distribution data," *Chem. Engng. Sci.,* **55**, 5907–5917 (2000).

Mavriplis, C., "Interdisciplinary CFD," *International Journal of Computational Fluid Dynamics*, **26**, 333–335 (2012).

Naumann, E.B. "Residence Time Theory," *Ind. Eng. Chem. Res*., **47**, 3752–3766 (2008).

Oliveira, A.G., G.P. da Silva, W. Geraldelli and K. Ropelato, "O crescimento do uso de fluidodinâmica computacional (CFD) nas tecnologias de destilação, FCC e coqueamento reatardado na engenharia básica de abastecimento da Petrobrás," *Petro & Química*, **343**, 222–226 (2012).

Patankar, S.V., *Numerical heat transfer and fluid flow,* 1. ed., Hemisphere Publishing Corporation, New York (1980).

Perry, R.H. and D.W. Green, *Chemical Engineer's Handbook,* 7th Edition, McGraw-Hill (1997).

Platvoet, E. and C. Baukal, "Process Burners," *Chem. Eng. Process*., **101**, 35-39 (2013).

Ranade, V.V., R.V. Chaudhari and P.R. Gunjal, *Trickle Bed Reactors*, Elsevier, (2011).

Ramdharee, S., E. Muzenda and M. Belaid, "A review of the equations of state and their applicability in phase equilibrium modeling," *Internation. Conference on Chemical and Environmental Engineering (ICCEE 2013)* (2013).

Sazhin, S.S., T. Kristyadi, W.A. Abdelghaffar and M.R. Heikal, "Models for fuel droplet heating and evaporation: Comparative analysis," *Fuel*, **85**, 1613–1630 (2006).

Vedovoto, J.J.M., R. Serfaty and A. da Silveira Neto, "Mathematical and Numerical Modeling of Turbulent Flows," *An. Acad. Bras. Ciênc.,* **87**, 1195-1232 (2015).

Zikanov, O., *Essential computational fluid dynamics*, 1. ed., John Wiley & Sons, New Jersey (2010).

Received: March 15, 2016.
Accepted: September 1, 2016.
Recommended by Subject Editor: Orlando Alfano