3D CFD simulation of Multi-phase flow separators

Zhiying Zhu*

School of petroleum engineering, China University of petroleum, Beijing, China

*Corresponding author e-mail: zhiying_2008@hotmail.com

Abstract. During the exploitation of natural gas, some water and sands are contained. It will be better to separate water and sands from natural gas to insure favourable transportation and storage. In this study, we use CFD to analyse the effect of multi-phase flow separator, whose detailed geometrical parameters are designed in advanced. VOF model and DPM are used here. From the results of CFD, we can draw a conclusion that separated effect of multi-phase flow achieves better results. No solid and water is carried out from gas outlet. CFD simulation provides an economical and efficient approach to shed more light on details of the flow behaviour.

1. Introduction

During the exploitation of natural gas, some water and sands are contained. And the existence of water and sands caused kinds of damages in the transportation and storage. For instance, water can reduce the delivery capacity of the pipeline [1]; sands can result in wearing down the facility of transportation, and sometimes blocking the outlet of the facility. Therefore, it is very important to remove the water and sands from natural gas.

The work process of separators is, firstly, natural gas with impurities flows into separate chamber. And then, because of gravity and/or centrifugal force, liquid and solid particles settle down to the bottom of chamber. Natural gas without impurities flows out from the outlet [2]. However, the complex flow behavior in the separator such as gas carry-under and efficiency should be considered also. Computational fluid dynamic (CFD) simulation provides an economical and efficient approach to shed more light on details of the flow behavior. Commonly, there are two types of separator, vertical type [3] and horizontal type [4]. Because we study natural gas with less water, and desand is a key need, we consider vertical type here. In this study, we design a separator by detailed geometrical parameters; utilize CFD software-Fluent to simulate three-phase flow in the separator. And analyze whether sand or water are carried out from the outlet with natural gas or not.

2. Modelling

The separator is shown schematically in Figure 1 and detailed geometrical parameters and explanations are illustrated in Table 1.

The part grid configuration is shown in Figure 2. Kinds of grid scheme were tested and we chose one of them here. Each circumference in z direction is set by interval count equals 60. And then apply Tet/hybrid scheme to the whole volume. Grid refinement study was conducted to obtain grid independent solution. The study indicated that a large number of grid points were required to obtain accurate solution for this 3D case. Other grids results are ignored in this paper.
Figure 1 Schematic of the separator configuration.

Table 1. Detailed geometrically parameters (units: mm) and explanations.

| H① | H② | H③ | H④ | H⑤ | D⑥ | D⑦ | A    | B    | C    |
|-----|-----|-----|-----|-----|-----|-----|------|------|------|
| 50  | 200 | 100 | 600 | 50  | 101.6| 30  | inlet| Outlet| Outlet|

(H means height, D means inner diameter. The inner diameter of A and B is 20.)
3. CFD simulation
Parameters information is as follows: Multi-phase flow in inlet is consist of natural gas, water and sands. Natural gas flow rate is 300,000m3/d, density is 1.293kg/m3 and viscosity is 0.0187mPa·s (standard condition, an atmospheric pressure, 20℃). Water flow rate is 60 m3/d and density is 1,000 kg/m3. Sand flow rate is 5-10 m3/d, density is 2700 kg/m3, thermal conductivity is 0.6W/m·K and specific heat is 900J/Kg·K. There are three kinds of sand with unique particle diameter, they are 0.15mm, 0.42mm and 0.21mm, and the proportion is 7%, 80% and 13% respectively. Inlet pressure is 32MPa, temperature is 90. B outlet pressure is 31.5-31.8MPa. Multi-phase flow contains not only fluid but also solid in this study, therefore, we solve the problem of gas and water firstly and then add solid to the gas-liquid condition. Gas and water are also two-phase, we use VOF model to solve it. And then we utility DPM model to analyze motion track of discrete phase-sand.

4. Result and discussion
Figure 3 and Figure 4 show velocity contours on part separator and velocity contours through the cross section of the inlet center in z direction. We can see that the fastest velocity appears in the inlet. And in Figure 4 we know that it is faster in near-wall region than far-wall region. This can help solid and liquid drop towards the bottom of the separator.
Figure 3 The velocity contours (part separator)

Figure 4 Velocity contours through the cross section of the inlet center in z direction

Figure 5 is the DPM result, it shows the trail of three kinds of sand particles. We can find that the bigger sands moves toward the bottom of the separator, the smallest particles move toward the top of the separator first and then get to the bottom, too. None of them flows out from gas outlet. Separated effect of multi-phase flow can achieve better results.
5. Conclusion
We can draw a conclusion that designed separator perform well in CFD simulation. No solid and water is carried out from gas outlet. CFD simulation provides an economical and efficient approach to shed more light on details of the flow behavior. We don’t need to make a real object and check again and again and make some changes. We can firstly do CFD simulation, and if the model run well, and then make a real object. CFD cannot work without real object experiment although.

Here, we use designed parameters, we can optimize the parameter in the future.

Acknowledgments
The research reported herein is partly supported by China University of Petroleum (Beijing) (grants No. ZX20150195). These financial supports are gratefully acknowledged.

References
[1] Chuang Wen, Xuewen Cao, Yan Yang, Effects of operating parameters on flow characteristics of natural gas in supersonic separators, Offshore Technology Conference (2013), OTC 24086.
[2] Liu Chao, Development of natural gas separator, Agricultural science and technology and equipment, No.227,2013, pp 29-30. (in chinese)
[3] Ferhat M.Erdal, Siamach A, Shirazi, Ovadia Shoham, Gene E.Kouba, CFD simulation of single-phase and two-phase flow in gas-liquid cylindrical cyclone separators, SPE-36645-PA, 1996.
[4] Seyed Ehsan Rafiee, M.M.Sadeghiazad, Experimental and 3D CFD analysis on optimization of geometrical parameters of parallel vortex tube cyclone separator, Aerospace science and technology, 63 (2017), pp 110-122.