2004

The Oil Vessel Structure Optimization by the use of CFD in the Oil Injection Twin-Screw Compressor

Gang Cheng
Fu Sheng Industrial (Shanghai) Co.

Li Yung Yan
Fu Sheng Industrial (Shanghai) Co.

Hua Zhou
Institute of Applied Mathematics and Mechanics Shanghai University

THE OIL VESSEL STRUCTURE OPTIMIZATION BY THE USE OF CFD IN THE OIL INJUECTION TWIN-SCREW COMPRESSOR

Cheng Gang[1], **Yan Li-yung**[2], **Zhou Hua**[3]

[1]FuSheng Industrial (Shanghai) Co., Ltd.
[1]No.28 mingyi Road, Xinqiao, Songjiang, Shanghai, P.R. China. 201612
Tel : 86-21-57686868(EXT396) Fax : 86-21-57686688
E-mail: Chenggang@fusheng-china.com

[2] FuSheng Industrial (Shanghai) Co.,Ltd.
[2]No.28 mingyi Road, Xinqiao, Songjiang, Shanghai, P.R. China. 201612
Tel : 86-21-57686868(EXT370) Fax : 86-21-57686688
E-mail: emerson@fusheng-china.com

[3] Institute Of Applied Mathematics and Mechanics Shanghai University.
[3]No.149, Yangchang Road, Shanghai, P.R. China. 200072
Tel : 86-21-56333085 Fax : 86-21-56333287
E-mail:zhouhua@mail.china.com

ABSTRACT

Oil consumption is a key economical factor in oil injection twin-screw air compressor. This paper presents the optimization of one kind of horizontal oil vessel to raise the first separation efficiency between oil and air by using CFD (computational fluid dynamics). The $k-e$ model and discrete phase model are used during the simulation. Experimental data are provided to modify relative parameters to get the most accurate model then, by which, a more efficient oil vessel with improved structure will be gotten and some conclusions will be concluded.

1. INTRODUCTION

The amount of residual oil content is an important quality in the oil injection screw air compressor, especially for environment and some special industries. For example, little oil would be consumed for an oil injection screw compressor to run hours with the residual oil content less than 2ppm(part per million) level. Developing low residual oil content and oil free air compressors becomes the new trend. With the development of computer technology, recently the application of computational fluid dynamics (CFD) becomes more and more widely in industry. This paper introduces one optimization about the structure of oil vessel in twin-screw compressor. The multiphase model of software FLUENT is used to simulate the flow field in the oil tank, and by analyzing the simulation results and experimental data, a new structure is designed and some conclusions are concluded.

2. CONSTRUCTION FOR SIMULATION

2.1 Solid Model For The Vessel

In this paper, an oil injection twin -screw air compressor is analyzed. The original structure of oil barrel is shown as figure 1.

The oil vessel mainly includes three parts, inlet pipe, oil barrel and outlet. The inlet pipe is a bended pipe by which the air-oil mixture could be injected to the oil barrel, then an injection flow would be formed between the pipe end and wall. The oil vessel is the main body, with some certain high level oil deposited at the bottom and absorbed by oil inlet pipe for cyclic using. There are two baffles at the two sides of the inlet pipe in the oil vessel. The original shape of outlet is circular with an arc cut at the right side to extend the path.

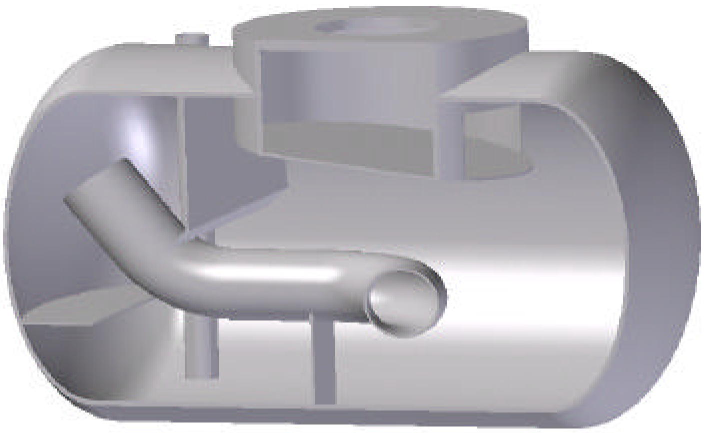

fig 1 solid model for the original oil vessel

Unstructured mesh are adopted in this calculation of flow field. Firstly, solid model should be constructed by one three-dimensional software , then ,it will be saved as *.stl format for gambit to mesh the whole body. The unstructured mesh is shown as followed,

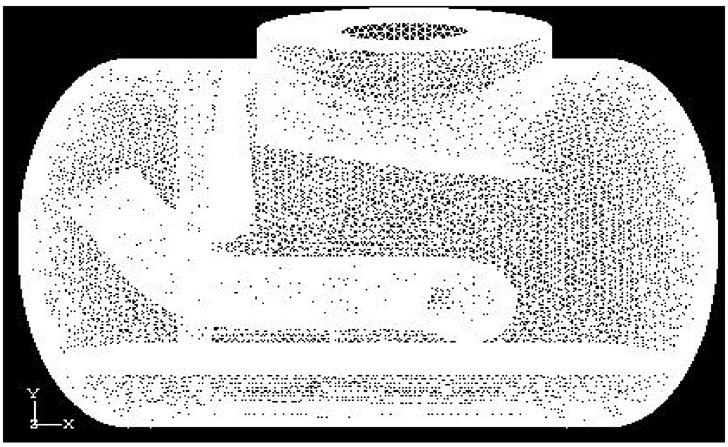

fig 2 unstructured mesh of the original oil vessel

2.2 Physical Model

The mixture in the oil vessel is main ly composed of gaseous air, oil and droplets of oil, and its temperature is about 90 degrees centigrade. This mixture would have two separation processes between the inlet and the outlet. The first separation process mainly gets rid of large oil droplets whose diameter is more than 10 um due to collision or cyclone principle, and the second separation process removes small oil droplets by special filter. Here only the first separation is discussed by the optimization of the tank.

2.3 Multiphase Model and Parameters

The commercial software FLUENT is used to simulate the flow field in the oil vessel. About this problem, because the oil droplets has a low volume fraction which is less than 10%, the author assumes that oil droplet is sufficiently dilute and its particle-particle interactions and the effects of the particle volume fraction on the air phase are negligible. The steady-particle Lagrangian discrete phase model is suitable for flows in which particle streams are injected into a continuous phase flow with a well-defined entrance and exit condition. So discrete phase model is adopted in the calculation.

Relative data show the diameter range of oil droplets in the tank is mainly 1 to 50 um, then a certain average value and distribution model are assumed for this simulation by trial method. The density of air is 8.4406 kg/m3 at 0.7MPa and 6.029 kg/m3 at 0.5MPa, for oil the density is about 870 kg/m3. The discrete phase particles (oil droplets) are taken as inert. To get the simulation , the standard $k-e$ turbulent model should be applied, the inlet boundary condition should be set as mass flow inlet and outlet should be set as pressure outlet for the better convergence, the other boundary parameters should be set as experimental data and the implicit formulation is used.

3. SIMULATION RESULTS

One section of velocity vector in the oil vessel is shown in figure 3 and the track curve of oil droplets are shown in figure 4,

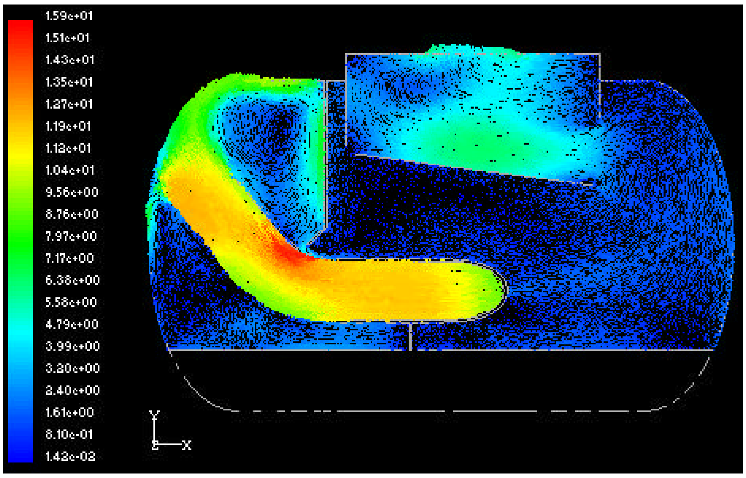

figure 3 velocity vector in the oil vessel (m/s)

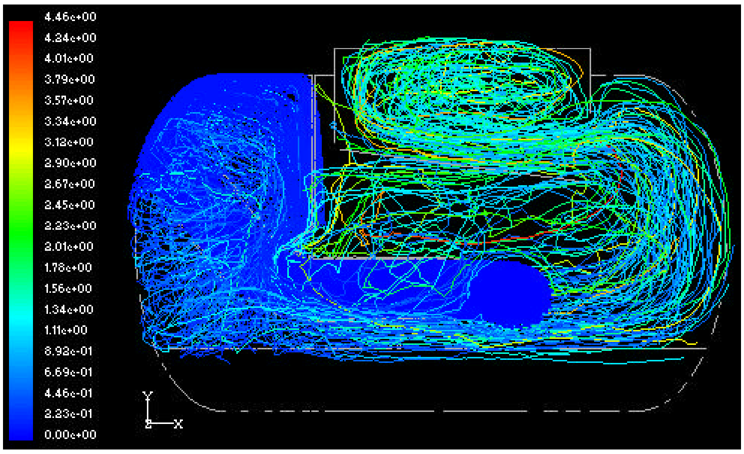

figure 4 track curve of oil droplets in the oil vessel (s)

The motion of the mixture in the oil tank can be seen from the above two figures. The air-oil mixture goes through the bended pipe and will be injected to the tank for separation, and somecollision occurs in the inner wall and sequentially large oil droplets are formed flowing to the bottom The separation between air and oil happens after the collision then eddies are formed flowing from the left of the baffle to the other side. Circuitous flow occurs before the mixture flows to the second part. Most of the mixture enters the second part and forms eddies going through the outlet to the separator. A little fraction of the mixture goes round in the tank and then mixes with the next mixture to the outlet. The section velocity vector of the outlet can be seen in figure 5

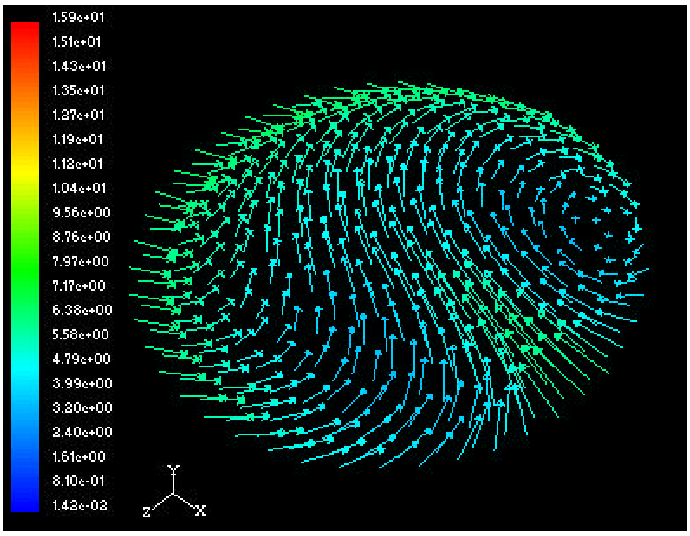

figure 5 velocity vector at the outlet (m/s)

Figure 5 shows the velocity at the outlet is not uniform, ranging from 3.6 m/s to 6.2 m/s, which will bring some negative effect on the following separation process.

Simulation results indicate the separation efficiency for this barrel is 99.975% at normal working condition when pressure is 0.7Mpa. The experimental datum is 99.980% , which is calculated by measuring the amount of oil through the scavenger line. Other working conditions indicate the simulation results and the experimental values are quite close, which proves the simulation by CFD on this problem is credible.

4. IMPROVED DESIGN

Considering the characteristics of flow field in the original oil vessel, a new tank is designed as figure 6

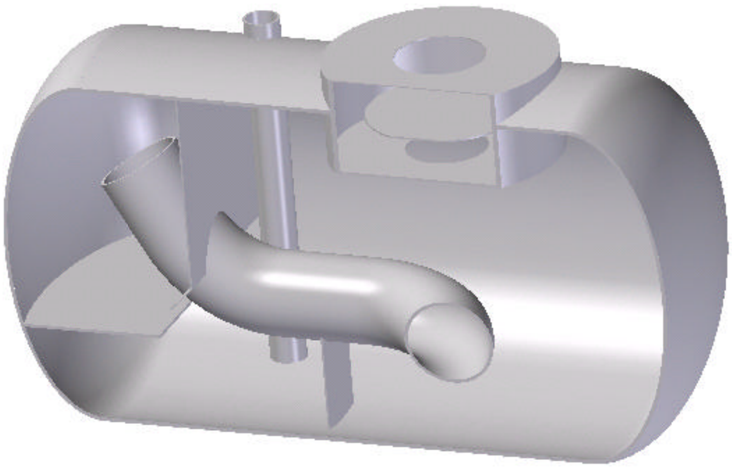

figure 6 solid model for new oil vessel

The section of velocity vector in the new tank is shown in figure 7 and the track curve of oil droplets are shown in figure 8 as below, and the velocity vector at the outlet can be seen from figure 9.

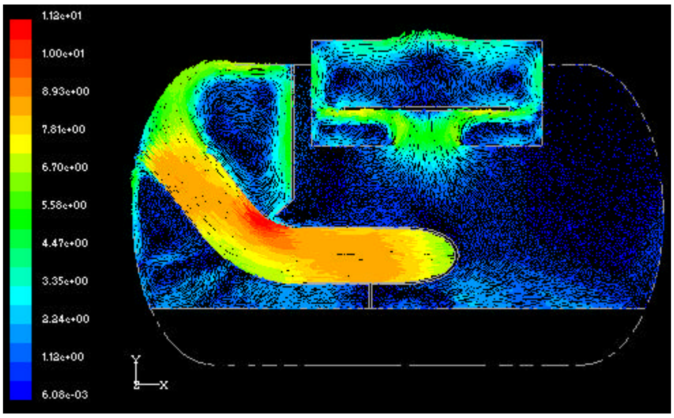

figure 7 velocity vector in the new oil vessel (m/s)

Figure 7 tells the second barrel has two collision processes and most of the oil droplets are gathered together to the bottom. It can be seen from Figure 9 that the velocity at the outlet is quite uniform about 5.0 m/s, which is good for the following separation process. The simulation results indicate the separation efficiency for the new barrel is about 99.995% with the same working condition, which is much better than the original one.

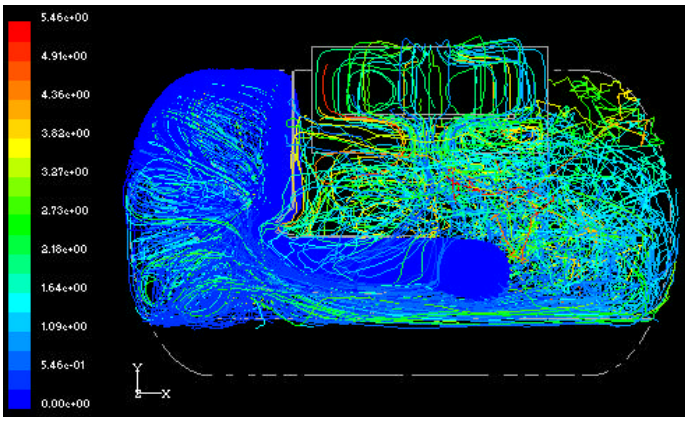

figure 8 track curve of oil droplets in the new oil vessel (m/s)

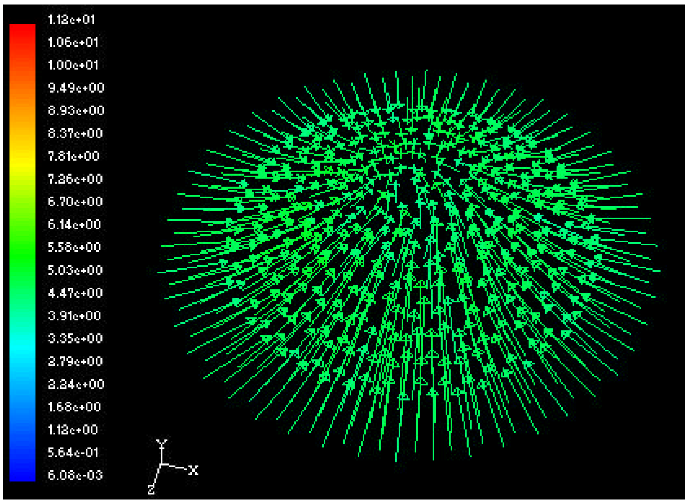

figure 9 velocity vector at the outlet in the new vessel (s)

5. CONSLUSIONS

Through this optimization of oil tank, CFD as a tool can be used to simulate the flow field in multiphase condition and to aid design. The precision of the results is sensitive to meshes, selected model and relevant parameters. The movement of particles can be analyzed from those flow field, then the results can direct and improve designs. Compared to traditional way, it can speed up the design process and bring visual analysis, and further more it is economical. In fact, experiments are absolutely essential to check and evaluate the final results.

REFERENCE

FLUENT User's Guide, Chapter 19, Discrete Phase Models.
Zhang Yu -ying, 2003, The design of oil-air separator and its core in oil injected screw compressor, COMPRESSOR TECHNOLOGY, vol. 2, p25.

ACKNOWLEDGMENT

Much thanks to Mr. Sun Wei-ming of Institute Of Applied Mathematics and Mechanics Shanghai University for his contribution to the simulation work., and thanks to FuSheng Company for providing much support on the research.