Transient simulation in interior flow field of lobe pump

X H Sang¹, Y B Li², Q W Meng³, H Shen¹ and K Jia¹

¹ School of Mechanical and Electrical Engineering, Lanzhou University of Technology, Lanzhou, 730050, Gansu, China.
² School of Energy and Power Engineering, Lanzhou University of Technology, Lanzhou, Gansu 730050, China.
³ Jinan UK Metering Flow Machinery Co., Ltd, Jinan, Shandong 250021, China.

E-mail: liyibin58@163.com

Abstract. The subject of this paper is mainly focused on the development and control of the double folium and trifolium lobe pump profiles by using the principle of involute engagement and use the CAD to get the accurately involute profile. Then using the standard k-ε turbulence model and PISO algorithm based on CFD software FLUENT. The dynamic mesh and UDF technology is introduced to transient simulation of the interior flow field inside a lobe pump, and the variation of interior flow field under the condition of the lobe rotating is analyzed. It also analyse the influence which produced by the differ in lobes, and then reveal which lobe is the best. The results show that dynamics variation of the interior flow field is easy to be obtained by dynamic mesh technology and the distribution of its pressure and velocity. Because of the small gaps existing between the rotors and pump case, the higher pressure area will flow into the lower area though the small gaps which cause the working area keep with higher pressure all the time. Both of the double folium and trifolium are existing the vortex during the rotting time and its position, size and shape changes all the time. The vortexes even get disappeared in a circle period and there are more vortexes in double folium lobe pump. The velocity and pressure pulsation of trifolium pump are lower than that of the double folium.

1. Introduction
The lobe pump belongs to a rotary pump, it is suitable for transporting of high viscosity, high concentration and any medium which containing particles. The lobe pump include pump body, rotors, pump bonnet, seal gland, cog-wheel gearing and driving gear device. there into, the two rotors clench the teeth in opposite direction and then the fluid is suctioned in the rotor cavity, it will gain energy via the rotor cavity and this energy will be converted to velocity near the outlet, so the pump gets the pressure and the rate of flow.

However, in China, due to the poor design theory of lobe pump, technology protection of foreign company and other negative factors, the research datum in this field is relatively rare. The main running characteristics and structure of lobe pump itself make it difficult to get the real variation of interior performance parameters through the experimental tools.

In recent years, with the wide application of CFD, it has became an effective research methods on kinds of numerical simulation. such as Dumont K et al. using dynamic mesh technique in Fluent to calculate the internal flow field of rotary pump [1]. Riemslagh Using the Lagrange- Euler finite volume
method to calculate the internal fluid flow characteristics of lobe pump and gear pump[2]. Jiang Fan, Chen Weiping et al. Using dynamic mesh technique to calculate the internal flow field of gear pump and centrifugal pump [3]. In this article, the standard k-ε turbulence model, PISO algorithm, the moving grid [3] and UDF technology is used to analyze the pressure distribution in the flow field inside the pump and also the velocity distribution of inlet in different time while the rotor is rotating.

2. Confirming the Lobe Pump Rotor Profiles
In fact, the lobe pump rotor profile is the external contour line (or tooth outline) of transverse section. Generally, the rotor profile of lobe pump has three models there are the circular arc, involute and cycloid respectively. Among the three models, the involute profile is easily to get higher precision and it also has higher volumetric efficiency. The following introduce rotor is the involute model and there always exist a small gap between the two rotors. In most cases, the value of small gap is decided by working experience [4,5].

According to the involute theory, we can get the equations of the double folium and trifolium lobe pump profiles, the equations of profiles can then be expressed as equation (1) and equation(2):

\[
x = r [\cos(\alpha + \varphi) + (\varphi + \tan \beta) \sin(\alpha + \varphi)]
\]
\[
y = r [\cos(\alpha + \varphi) - (\varphi + \tan \beta) \cos(\alpha + \varphi)]
\]

(1)

\[
x = r [(\cos \varphi + \pi / 180 \varphi \sin \varphi) \cos \alpha - (\sin \varphi - \pi / 180 \varphi \cos \varphi) \sin \alpha]
\]
\[
y = r [(\cos \varphi + \pi / 180 \varphi \sin \varphi) \sin \alpha + (\sin \varphi - \pi / 180 \varphi \cos \varphi) \cos \alpha]
\]

(2)

Table 1. The parameters of two type involute profiles

| Type      | Pitch radius R (mm) | Pressure angle \(\alpha\) (\(^\circ\)) | Ground radius r (mm) | The range of involute angle \(\varphi\) (\(^\circ\)) |
|-----------|---------------------|-----------------------------------------|-----------------------|-----------------------------------------------|
| Double folium | 40                  | 45                                      | 30                    | 0\(^\circ\)~90\(^\circ\)                    |
| Trifolium  | 131                 | 52.785                                  | 36.288                | 15.92\(^\circ\)~75.92\(^\circ\)             |

So, we can draw it in CAD according to the above equations and table 1.As shown in Figure 1. are double folium and trifolium lobe involute profiles.

![Figure 1. Double and trifolium lobe involute profile.](image)

3. Numerical Calculation Method for Lobe pump

3.1. Simplified the Geometry
As we known the numeric simulation needs higher computer configuration, especially the 3D model, while the 2D is enough in flow field analysis. thus the 2D geometry is used in following simulation.

The 2D geometry contains three parts: inlet part, outlet part and rotors rotting part. Both of the inlet and outlet width (length) are 40mm (120mm), the gap between rotors and the interior wall are all 0.4mm.

3.2. Numerical calculation method

3.2.1. Grid division. In order to get the fine grids we divide the geometry into three parts, which are named working area of the lobe pump, entrance passageway and outlet passageway. In the Gambit, grid type selects the triangle (Tri). Division method selects the unstructured grid (Pave). Interval size of grid selects 0.25mm in the working area, and selects 0.5mm in the area of two passageways. For the inward and outward boundary choose the pressure of inlet and outlet. defining the profiles of the two rotors as wall. The double folium lobe pump has 587656 triangle cells and 293827 nodes. The three blades lobe pump has 497028 triangle cells and 248513 nodes.

3.2.2. The control equations and discrete methods. The standard K-Epsilon model is used:

\[
\begin{align*}
\rho \frac{d k}{d t} &= \nabla \cdot \left[ \left( \mu + \mu_t \right) \nabla k \right] + G_k + G_b + \rho \varepsilon - Y_m \\
\rho \frac{d \varepsilon}{d t} &= \nabla \cdot \left[ \left( \mu + \mu_t \right) \nabla \varepsilon \right] + C_{1k} \frac{\varepsilon}{k} (G_k + C_3 \varepsilon G_b) - C_{2k} \rho \varepsilon^2
\end{align*}
\]

while \( C_1 = \max \left[ \frac{0.43 \eta}{\eta + 5} \right], \eta = Sk / \varepsilon \).

where \( C_2 \) and \( C_{1k} \) are constant; \( \sigma_k \) represents turbulent kinetic energy; \( \sigma_\varepsilon \) represents turbulent Prandtl number of turbulent kinetic energy dissipation rate. In Fluent, as the default constant, \( C_{1k}=1.44, C_2=1.9, \sigma_k=1.0, \sigma_\varepsilon=1.2 \).

Selects version as 2ddp and chose mode as full simulation, time as unsteady, Selects viscous model as standard k-\( \varepsilon \) turbulence and PISO algorithm. selects discretization as pressure, others selects second order upwind.

Others default and referring the detailed setting to the reference [6-8].

3.2.3. The boundary conditions and initial conditions. Setting inlet as pressure-let and outlet as pressure-let too, the profile of two rotors as wall, the rotating speed is 300r/min. Selects water-liquid from the fluent database materials. its density is 998.2kg/m\(^3\), viscosity is 0.01003N \( \cdot \) s/m\(^2\).

Add one pressure and velocity monitor around inlet and outlet separately. Then sit the autosave and data file frequency as 250.

Using the UDF function to make the rotors rotate in the opposite direction in FLUENT.

The C file as follows:

```c
#include<stdio.h>
#include"udf.h"

DEFINE_CG_MOTION(rotting_left, dt, vel, omega, time, dtime)
{
    omega[2]=10*3.1415926;
}

DEFINE_CG_MOTION(rotting_right, dt, vel, omega, time, dtime)
{
    omega[2]=10*3.1415926;
}
```
4. The Simulation results and analysis

In order to analyse the dynamic variation of the interior flow field, now we emphasize on the distribution of interior pressure and velocity in different time.

Figure 2. (a) - (d) indicates the interior static pressure distribution of double folium and trifolium lobe pump rotates a circle period from time 0.025s to 0.1s. As depicted in the pictures, there is no significant change in color around the inlet, explains the pressure there is lower and well-distributed. The value equates to the initial pressure, however, the pressure of the working area changes significantly which explains that pressure is not evenly distributed, the gradient of pressure is larger everywhere. It form the lower pressure area around the small gap between two rotors and the driving rotor drags the lower pressure area into the working area. It also showed the minimum pressure consist in the gaps of two rotors and Pump wall. Because of the existing gaps, the higher pressure area pour into the lower area with high backflow, it convert the pressure into kinetic energy and the pressure get reduced.
Figure 2. Contours of static pressure (pascal) with different foliums.

Figure 3. (a) - (d) are the interior velocity vector of double folium and trifolium lobe pump rotates a circle period from time 0.025s to 0.1s. owing to the rotors rotation extrusion and the gaps, a large number of vortex has arisen when high backflow into the lower pressure area. the vortexes will be produced, developed and disappeared in a circle period as a result of its position, size and shape Changes with the rotation. We can draw a conclusion both of the size and quantity of trifolium rotors are bigger than the double folium rotors. so, the internal flow field of trifolium is more stable than the double folium.

Figure 3. The streamline chart with different foliums.

Figure 4. and Figure 5. are the inlet pressure and velocity vs. Different time with double folium and trifolium. According to the two pictures, at the beginning of the pump working both of the velocity and pressure fluctuate strongly, however, after the up time the period repeated very well. Fig.6 and Fig.7 indicate the velocity pulsation and pressure of trifolium pump are lower than the double folium.
5. Conclusions
In the present paper the double folium and trifolium lobe pump are presented. The involute profiles were designed using CAD tools and simulated using software FLUENT which emphasis on the interior pressure and velocity vs the different period time. Based on the obtained results the following can be concluded:

1) Because of the small gaps existing between the rotors and pump case, the higher pressure area will flow into the lower area though the small gaps which cause the working area keep with higher pressure all the time.

2) Both of the double folium and trifolium are existing the vortex during the rotting time and its position, size and shape changes all the time. The vortexes even get disappeared in a circle period. There are more vortexes in double folium lobe pump.

3) The velocity pulsation and pressure of trifolium pump are lower than the double folium. Further work will focus mostly on the three-dimensional lobe pump, detailed investigation of the different speed of revolution and gaps explaining the influences on interior flow distribution.

References
[1] Dumont K and VandeV J 2004 J.of Computational and Applied Mathematics 168(1-2) 491-9
[2] Riemslagh, Vierendeels and Dick 2000 Applied Numerical Mathematics 32(4) 419-33
[3] Jiang F, Chen W P and Wang Y 2007 J. Fluid Machinery 35(7) 20-24
[4] Zhang YF, Yang F L, Zhou W C and Xing J L 1999 T. J. of Zheng Zhou University of Technology 20(3) 87-89
[5] Tang S H, Zheng J C 2007 China Oils and Fats 32(5) 65-67
[6] Liu Z X 2004 Fluid Machinery 32(10) 1-4
[7] Wang F J 2004 Computational Fluid Dynamic Analysis of the Principle and Application of CFD Software (Beijing: Tsinghua University Press)
[8] Jiang F and Huang P 2008 Fluent Advanced Application and Analysis of Example (Beijing: Tsinghua University Press)