Numerical study and experimental validation of a Roots blower with backflow design

Shu-Kai Sun\textsuperscript{a,b}, Xiao-Han Jia\textsuperscript{a}, Lin-Fen Xing\textsuperscript{a} and Xue-Yuan Peng\textsuperscript{a}

\textsuperscript{a}School of Energy and Power Engineering, Xi’an Jiaotong University, Xi’an, People’s Republic of China; \textsuperscript{b}School of Resources Environmental & Chemical Engineering, Nanchang University, Nanchang, People’s Republic of China

ABSTRACT

A three-dimensional computational fluid dynamics (CFD) model of a Roots blower with a backflow design was established to analyse the effects of the backflow on the Roots blower’s performance. A prototype of Roots blower with a backflow design was manufactured to validate the CFD model through the pressure distribution and the mass flow rate. The results showed that the proposed CFD model agreed well with the experimental data. The effects of the sizes and directions of the backflow passage on the Roots blower’s performance were then investigated using the validated CFD model. It was found that under a properly sized backflow passage, the pressure pulsation and the shaft power can be decreased by 80% and 13%, respectively; however, the mass flow rate was reduced by 12% under the same size of backflow passage. Although the direction of the backflow passage affected the shaft power, it had no effect on the mass flow rate or pressure pulsation. The shaft power consumption of a Roots blower with a vertical backflow was 4% lower than a horizontal backflow.

ARTICLE HISTORY

Received 22 May 2017
Accepted 15 December 2017

KEYWORDS

Roots blower; backflow; numerical study; dynamic mesh

1. Introduction

The Roots blower is a type of gas transport machine comprising a pair of meshed but non-contacting rotors in the cylinder. The rotors rotate in opposite directions and are driven by two identical timing gears. The fluid is transferred from the inlet side to the outlet side when the rotors rotate as shown in Figure 1. Roots blowers are widely used in the pharmaceutical industry, chemical industry, and sewage treatment because of their high reliability, low cost and easy maintenance. Roots blowers can be applied to numerous devices such as middle-pressure blowers, low- and middle-vacuum pumps, and hydraulic pumps. In recent years, the Roots blowers were used in mechanical vapour recompression systems to increase the pressure and temperature of the vapour and to recycle hydrogen as well as supply the compressed air in hydrogen fuel cells.

The development of Roots blowers primarily involves designing the profile of the rotor, which affects the meshing and performance of the blower. Tong and Yang (2000, 2005) and Yang and Tong (2002) presented a method for designing a new profile and established a function for analysing the main factors affecting the efficiency of high-sealing Roots blowers. Wang, Fong, and Fang (2002) discussed the constraints for the tooth profile of five-arc Roots vacuum pumps. Mimmi and Pennacchi (1999) proposed an explicit three-dimensional (3D) analytical model of the screw rotor considering all the surfaces which form the rotor and their parametric expressions. Litvin and Feng (1996) proposed an approach for the design and generation of the planar cycloidal gearings for screw Roots blowers and developed an improved design to avoid the surface singularity. Kang, Vu, and Hsu (2012) developed a Roots blower with an epicycloid curve, which demonstrated a better performance over circular ones. They also found that multi-lobe pumps did not exhibit a higher performance than two-lobe pumps but provided a more stable output. Hwang and Hsieh (2006) and Hsieh and Hwang (2007) designed smooth trochoid ratio profiles by high-order polynomials which improved the sealing performance and volumetric efficiency of the Roots blower. Burmistrov, Belyaev, Ossipov, Fomina, and Khannanov (2001) proposed an angular-coefficient method for calculating the channel conductance in Roots blowers. Yao, Ye, Dai, and Cai (2005) proposed a new type profile which could create a higher air flow and a lower peak of the blower pressure compared to the conventional straight tooth. Valdes, Barthod, and Perron (1999) presented a calculation method which was able to accurately calculate the inside leakage for...
Many researchers analysed the complex flow phenomena inside positive-displacement machines such as screw compressors and Roots blowers by CFD technology, as it showed a high accuracy in engineering practice. A new type of numerical grid generation for twin screw rotors was proposed based on rack generation in order to develop the 3D CFD model of the screw compressor (Arjeneh, Kovacevic, Rane, Manolis, & Stosic, 2015; Broatch et al., 2015; Kovacevic & Smith, 2002; Kovacevic, Stosic, & Smith, 2007; Stosic, Smith, Kovacevic, & Mujic, 2011) These studies improved the performance of the twin-screw compressor and expander by analysing the pressure, temperature, leakage, and velocity through CFD simulations. In addition, the complex flow field inside the cylinder of the Roots blowers was also investigated by CFD simulations in many studies. Li, Jia, Meng, Shen, and Sang (2013) analysed the effects of different pressure angles on the velocity and pressure of oil as well as the turbulent kinetic energy by 2D numerical modeling. Joshi, Blekhman, Felske, Lordi, and Mollenendorf (2006) analysed the clearance between the rotors, showing that the clearance remains constant for 96% of the angles of the rotation in a working period. They also investigated the leakage of the Roots blower through a quasi-steady state analysis. Hsieh and Deng (2015), and Hsieh and Zhou (2015) compared the differences in the mass flow rate and pressure between Roots blowers with screw type rotors and cylindrical rotors using the CFD solver PumpLinx. They also analysed how the rotor phase affects the flow rates and pulsation of a serial multi-stages and a parallel multi-stage Roots blower. Huang and Liu (2009) identified and studied the characteristics of flow in a positive displacement blower through the CFD simulation of an involute-type blower with three-lobe positive discharges containing an unsteady compressible flow.

A large amount of aerodynamic noise and air pulsation is often generated by Roots blowers because of their unique compression process which limits the application field of Roots blowers. The working principle of the Roots blower is shown in Figure 2. When the rotation angle is between 0° and 120°, the volume of the working chamber increases, and the air flows from the intake cavity to the working chamber. After 120°, the working chamber is disconnected from the intake cavity, and the moving chamber transfers air by the rotor rotation. When the rotor rotating angle reaches 180°, the working chamber meets with the exhaust cavity. The pressure in the working chamber is increased by the air with a higher pressure in the exhaust cavity, and this often causes the airflow impact and air pulsation. With rotor rotation over 180°, the volume of the working chamber starts to be reduced, which reaches 0 at the rotation angle of 300°. Ohtani and Iwamoto (1981) experimentally demonstrated the air backflow could effectively reduce the aerodynamic noise by reducing outflow pulsation. It is necessary to analyse how the pressure pulsation was affected by the backflow. Although the backflow design for the performance of Roots blowers can hardly be completed in the traditional way, the development of CFD and dynamic mesh technology provided a new way to examine how the backflow design affects the performance of a Roots blower during the working process.
In the present study, 3D CFD models of a Roots blower with and without a backflow design were established to analyse the inside flow field in their working process. A new type Roots blower identical to the simulation one was designed and manufactured. The flow rate and the pressure distributions in the cylinder and intake/exhaust cavities were measured to validate the result from simulations. Based on all the results from experiments and simulations, the performances of the Roots blowers that have and do not have a backflow design were investigated and compared. Finally, the effect of various backflow passage sizes and directions on the flow field were studied through the validated CFD models.

2. Numerical model

2.1. Structure of backflow design

Figure 3 shows the structure of a Roots blower with backflow design. There are eight holes symmetrically distributed on the upper and lower working chambers, through which the higher-pressure gas can afflux into the cylinder earlier. Four of these holes are horizontally located at the rotation angle of 120° while the other four are vertically distributed at the same rotation angle.

Each rotor in this Roots blower has three lobes, which creates six sections with a profile curve consisting of arc line, straight line, involute, straight line, and two arc lines successively (Peng, 2000). The cylinder has a diameter and length of 52.5 and 70 mm, respectively. The double rotors’ centre distance is 70 mm; the sizes of the clearances in the cylinder are summarised in Table 1.

| Clearances                                      | Size (mm) |
|------------------------------------------------|-----------|
| Between the two rotors                         | 0.14      |
| Between the rotor and the cylinder in the circumference direction | 0.07      |
| Between the rotors and the cylinder in the axial direction on the axis side | 0.03      |
| Between the rotors and the cylinder in the axial direction on the cover side | 0.11      |

The fluid field in the Roots blower, taking account of the effect of backflow, was investigated by CFD simulations, the geometry of which is shown in Figure 4. The geometry has the same size as the Roots blower, except for the clearances. In the CFD model, the clearances between the rotors and the cylinder in the axial direction are ignored because the dynamic mesh is hard to apply in them. The clearances between the rotor and the cylinder in the circumferential direction in the CFD model are increased to 0.15 mm in order to keep the total leakage area unchanged. A narrow pipe was set on the outlet piping throttle of the flow to make the pressure in the outlet piping reach the exhaust pressure, as shown in Figure 5; to obtain a higher precision of the pressure change in the exhaust cavity (Sun, Zhao, Jia and Peng, 2017).

2.2. Governing equations

The continuity equation, the energy conservation equation and the momentum equation comprise the governing equations. In this study, the simulation was carried out through Fluent (a commercial CFD software), where
the governing equations were as shown in Equations (1)–(3). (Fluent Inc., 2013):

\[
\frac{\partial p}{\partial t} + \nabla \cdot (\rho \mathbf{v}) = 0
\]  

(1)

\[
\frac{\partial (\rho E)}{\partial t} + \nabla \cdot (\rho \mathbf{v}(E + p)) = -\nabla \cdot \left( \sum_{j} h_{j} \mathbf{l}_{j} \right) + S_{h},
\]  

(2)

\[
\frac{\partial (\rho \mathbf{v})}{\partial t} + \nabla \cdot (\rho \mathbf{v} \mathbf{v}) = -\nabla p + \nabla \cdot \mathbf{f} + \rho \mathbf{g} + \mathbf{F}
\]  

(3)

In these equations, \( p \) is the static pressure, \( \mathbf{F} \) is the external body force, \( \rho \mathbf{g} \) is the gravitational body force, and \( \mathbf{f} \) is the stress tensor.

The turbulence equation is critical for the simulation of Roots blowers as the backflow causes high rotating shear flows in the blower’s working process. Separated flows are generated when the air flows across the top of the rotating rotor. Li, Zhang, Zhu, and Hu (2007) compared the simulation results of the Realizable \( k-\epsilon \) mode, the RNG \( k-\epsilon \) model and the standard \( k-\epsilon \) model for a centrifugal pump. It was found that the best simulation result was obtained by the Realizable \( k-\epsilon \) model. In consideration of the high speed of the lobe of the Roots blower and centrifugal pump, the Realizable \( k-\epsilon \) model was then chosen to work as the turbulence equation. The transport equations of the Realizable \( k-\epsilon \) turbulent model are as follows:

\[
\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}(\rho k u_j) = \frac{\partial}{\partial x_j}\left[\left(\mu + \mu_t\right)\frac{\partial k}{\partial x_j}\right] + G_{k_{v}} + G_{k_{b}} - \rho \epsilon - Y_{M} + S_{k}
\]  

(4)

\[
\frac{\partial}{\partial t}(\rho \epsilon) + \frac{\partial}{\partial x_j}(\rho \epsilon u_j) = \frac{\partial}{\partial x_j}\left[\left(\mu + \mu_t\right)\frac{\partial \epsilon}{\partial x_j}\right] + \rho C_{1_{s}} \epsilon - \rho C_{2_{s}} \frac{\epsilon^2}{k + \sqrt{\epsilon}} + C_{1_{v}} \frac{\epsilon}{k} C_{3_{v}} G_{k_{b}} + S_{\epsilon},
\]  

(5)

where

\[ C_1 = \max \left[ 0.43, \frac{\eta}{\eta + 5} \right], \eta = \frac{k}{\epsilon}, S = \sqrt{2S_{ij}S_{ij}}. \]

where \( C_{2}, C_{1_s}, \sigma_{\epsilon}, \sigma_{k} \) are constants with values of them being 1.9, 1.44, 1.2 and 1.0, respectively; \( Y_{M} \) is the effect of fluctuating dilatation on dissipation rate; \( G_{k_{b}} \) is the turbulence kinetic energy produced by buoyancy; and \( G_{k_{v}} \) is turbulence kinetic energy produced by average velocity gradients.

2.3. Boundary conditions

Both the inlet and outlet pressure were 0 kPa (gauge pressure). The inlet pressure was set to 0 kPa which means that suction happened under atmospheric pressure. The exhaust pressure was set at 50 kPa because the narrow pipe in front of the outlet buffer area throttled the flow and increased the pressure in the outlet piping. The size of the narrow pipe was determined using the trial calculation method. The interface method was used for connections between the cylinder and the backflow piping. Each interface could be set as a wall boundary condition, whereby the number of working backflow pipes was controlled. Air was set as the working fluid, which is assumed as an ideal gas.
Table 2. Mesh types and numbers of elements for the Roots blower with the backflow.

| Part               | Mesh type                | Number of elements |
|--------------------|--------------------------|--------------------|
| Inlet buffer area  | Hexahedral element       | 105,858            |
| Inlet piping       | Hexahedral element       | 61,520             |
| Cylinder           | Prism element            | 112,322            |
| Backflow piping    | Hexahedral and tetrahedral element | 104,445 |
| Outlet piping      | Hexahedral element       | 290,480            |
| Outlet buffer area | Hexahedral element       | 105,858            |
| Total              |                          | 780,483            |

2.4. Grid generation and dynamic mesh

Figure 5 shows the grids of the numerical simulation model. The mesh type and the number of elements for each part are summarised in Table 2.

Since the Roots blower works as an unsteady process, the shape of its inside flow field changes during its working period. Therefore, a dynamic mesh was required for the CFD simulation. The grids of the rotor wall were driven by a UDF (user-defining function), which rotated as rigid bodies. The spring-based smoothing and 2.5D local-remeshing dynamic mesh methods were used for simulations due to the constant deformation of the cylinder.

The time-step was determined as \( \Delta t = \frac{(\text{CFL} \cdot \Delta x)}{\lambda_{\text{max}}} \), which was set to \( 1 \times 10^{-5} \) s. The case was calculated by a computer with 32 cores, 2.6 GHz, and 128 Gb of operating memory. It took 109 h to reach convergence and another 56 h to complete the simulation.

2.5. The grid and time step independence study

The grid independence study was carried out to compare the results of the simulations obtained under different grids. As shown in Figure 6 (a), the difference in the results between the element number in the range from \( 7.8 \times 10^5 \) to \( 1.75 \times 10^6 \) was less than 1.5%. Therefore, it was concluded that the independent results of the grid could be confirmed in a grid of \( 7.8 \times 10^5 \) cells.

A time-step independence study was performed to compare results of the simulations obtained under different time-steps. Figure 6 (b) shows the results of the simulation with the element number of \( 7.8 \times 10^5 \) for meshes with different time-steps. The difference between the results of the simulation with different time-steps was less than 0.5%. It was therefore concluded that the independent results of the time-step could be confirmed in the time-step of \( 1 \times 10^{-5} \) s. When the time-step was longer than \( 1 \times 10^{-5} \) s, a negative grid was generated and the simulation was unable to process.

3. Experiment on a Roots blower

To verify the accuracy of the simulation, experimental testing equipment was established, as shown in Figure 7. This equipment consisted of inlet/outlet pipes, inlet/outlet valves, pressure gauges, backflow piping, a flow meter, and pressure sensors. The valves in the inlet/outlet piping were used to adjust the intake/exhaust pressure, with pressure gauges installed to measure pressures. A flowmeter was installed to measure the average flow rate. There were five pressure sensors on the cylinder, which are all located on the upper side as the shape of the two rotors was identical and the installation locations were symmetrical with a 60° phase difference. Three of these pressure sensors were used to obtain the variation of the pressure in the working chamber while the rest were installed to measure the pressure in the intake/exhaust cavity. When the rotor rotated, the working chamber passed through these five sensors in turn.

Figure 6. The grid and time step independency.
Figure 7. Experimental equipment.

The pressure variation of the working chamber during a working period was obtained by collecting the pressure data from all five sensors. The backflow holes were connected to the outlet piping by the backflow piping, where a valve was installed in each to conduct the operation of each backflow pipe separately.

The pressure variations in the cylinder and the flow rate were obtained through the aforementioned experimental equipment. Operating conditions in experiments were as follows. The rotation speed of the motor was 2,400 r/min. The intake/exhaust pressure was adjusted by valves on the inlet/outlet piping, which were the same as the simulation. The sampling rate of the pressure sensors and the measurement range of sensors were 10 kHz and 100–250 kPa (gauge pressure), respectively.

### 4. Results and discussion

#### 4.1. Validation of simulation results with experimental data

To verify the accuracy of the simulation results, two cases were chosen for comparison with the experimental one regarding the mass flow rate and pressure variations. The first case was a Roots blower without a backflow design. In this case, the wall boundary condition was used for all the connections between the cylinder and the backflow piping. In the experiment, all eight valves in the backflow pipes were closed. The second case was a Roots blower with a backflow design. The boundary conditions of the connections between the cylinder and the vertical backflow piping were changed to an interface in this case, and the valves in the vertical backflow pipes were opened, while the horizontal ones had the wall boundary feature and were closed.

The CFD simulation of the Roots blower is an unsteady case with an initial value. Calculations for a great many time-steps are required before the simulation reaches a steady state. One indicator of the simulation reaching steady state was whether the difference in the average mass flow during a working circle between the inlet and the outlet was reduced to less than 5%. Table 3 shows that the difference was 1.6% without backflow and 3.3% with backflow, suggesting that both the cases were periodically steady until the third working circle. The experimental results for the mass flow rate are also shown in Table 3. The results indicate that the pumping rate in the simulation was 5.3% lower than in the experiment for the Roots blower without the backflow design and 2.3% lower for the one with the backflow design. The difference was small; thus, the simulation results for the mass flow rate are accurate.

|                    | Experiment (kg/min) | CFD simulation (kg/min) |
|--------------------|---------------------|-------------------------|
| Roots blower without backflow | 0.885               | 0.8538                  | 0.9324 | 0.9024 |
| Roots blower with backflow     | 0.8538              | 0.8736                  | 0.9024 |
The pressure variations for the Roots blowers with and without the backflow design are shown in Figure 8. Both simulation cases agree well with the experimental results.

Figure 8 also shows that the pressure variations for the Roots blower with a backflow design were almost the same as those for the Roots blower without the backflow design in the range of 0° to 120°. This is because the working chamber is still disconnected from the backflow piping, so the backflow showed little impact. However, at angles larger than 120°, the difference in the pressure variation became enhanced. The working chamber of the blower without the backflow design was closed from 120° to 180°, so the high-pressure gas in the exhaust cavity could only leak into the working chamber through the clearances, which slowly increased the pressure in the working chamber. At angles larger than 180°, as

Figure 8. Variations in the pressure with respect to the angle of the rotor in a working cycle.

Figure 9. The comparison of the Roots blower with and without backflow on pressure field.
Figure 9 shows, the working chamber would open to the exhaust cavity; the air in the exhaust cavity reflowed to the working chamber driven by the pressure difference. The direction of the exhaust process and the rotor rotation was inverse to the reflow, which leads to air pulsation and impact. The pressure pulsation amplitude was about 30 kPa. Although the working chamber was separated from the intake cavity from 120° and disconnected from the exhaust cavity until 180° in the blower containing a backflow design, the pressure in the working chamber was increased by the high-pressure air influx through the backflow piping. In this manner, the pressure difference between exhaust cavity and the working chamber was reduced before the working chamber opened to the exhaust chamber at 180°. The flow impact was reduced because of the small pressure difference. Thus, the pressure pulsation in the exhaust cavity was only 20 kPa, which is less than that of the Roots blower having no backflow design. That the pulsation in the exhaust pressure could be reduced effectively by the backflow design was proved by both simulation and experiment.

4.2. Effect of backflow on pressure pulsation in the exhaust cavity

Figure 10 shows the frequency characteristics of the pressure pulsation in the exhaust cavity. It showed the fundamental frequency of the pressure pulsation was 300 Hz, which was six times more than the frequency of the rotor rotation, because there were six exhausting periods during the 360° rotor rotation. The backflow design can reduce the amplitude of both the fundamental frequency and multiple frequencies.

To determine the effect of the backflow passage area on the operation of the Roots blower, different areas were simulated. With specific backflow pipes closed or open, the effects of the backflow direction on the operation of the Roots blower were examined. Figure 11 shows the pressure difference between the exhaust cavity and the working chamber at 180° for different backflow passages. As shown in Figure 11, the cases of only horizontal backflow holes, only vertical backflow holes, and both horizontal and vertical backflow holes yielded a horizontal backflow, a vertical backflow, and both horizontal and vertical backflows, respectively. The pressure difference decreased with the increase of the backflow area until the pressure in the exhaust cavity and working chamber were equal. Then, the pressure in the chamber exceeded the exhaust pressure, but it increased slowly. This is because the backflow came from the outlet piping, and pressure remained in the outlet piping. Figure 11 also shows that the variation of the pressure difference was independent of the backflow direction.

Figure 12 shows the variation of the pressure-pulsation amplitude in the exhaust cavity with respect to the pressure difference. A larger pressure difference caused a larger amplitude of the exhaust pressure pulsation. The pressure pulsation was independent of the backflow direction. A proper size of the backflow passage could lower the pressure pulsation in amplitude by 820%.
4.3. Effect of backflow on mass flow rate

How the pressure difference changes with the variation in the mass flow rate is shown in Figure 13. As the pressure difference decreased, the mass flow rate also decreased. This was because the pressure was increased due to a small difference in pressure in the working chamber, which led to a greater air leakage from the working chamber to the intake cavity and reduced the mass flow rate. However, this reduction was slow. The mass flow rate decreased by approximately 12% – from 0.93 to 0.84 – when the pressure difference decreased from 40 to 0 kPa. Figure 13 shows that the variation of the mass flow rate was independent of the backflow passage direction.

4.4. Effect of backflow on shaft power

To determine how the shaft power changed in the working circle, the torque caused by flow force of the upper rotor in the Roots blower was monitored during the CFD calculations. Figure 14 compares the Roots blowers with and without a backflow. The vertical backflow and the horizontal backflow were the same size. The backflow could significantly reduce the peak shaft torque. The torque of the rotor showed a maximum value at 60° (180°) in the Roots blower without backflow, because the working chamber connects to the exhaust cavity after 180°, which reduces the pressure difference on the lobe and reduces the torque of the rotor and shaft. The pressure difference between the two sides of the lobes in a Roots blower featuring a backflow design was relatively low compared to that which had no backflow feature. This is why the backflow could significantly reduce the peak shaft torque. Considering the rotation rate and the downer rotor, the average shaft powers are shown in Table 4. The results show the vertical backflow yielded the least shaft power, which was 4.1% lower than the shaft power for the horizontal backflow and 13% lower than that for no backflow. This is because the vertical backflow direction agreed with the rotation direction of the rotors.

5. Conclusion

Based on the numerical simulation and experimental studies above, the following conclusions can be drawn.
(1) The pressure pulsation in the exhaust cavity was reduced by the backflow. As the area of the backflow increased, the pressure pulsation decreased until the pressure in the working chamber reached the same value of the exhaust pressure at a rotor rotation angle of 180°. At this time, the pressure pulsation in the exhaust cavity was reduced by 80%. Then, as the area increased, the pressure pulsation increased slightly. The pressure pulsation was independent of the backflow direction.

(2) The backflow reduced the Roots blower’s mass flow rate because it increased the leakage. However, the amplitude reduction of the mass flow rate was not large (approximately 3–12%, depending on the size of the backflow holes). It was helpful for reducing the pressure pulsation even though the mass flow rate was reduced. The mass flow rate was independent of the backflow direction.

(3) The backflow could reduce the power consumption of the shaft in the Roots blower. The power consumption of the shaft in the Roots blower containing a vertical backflow was 4% lower than the blower which had a horizontal backflow because the backflow direction of the first pump agreed with the rotation direction of the rotors.

This study mainly considered the effect of the backflow passages on the pressure pulsation, mass flow rate and power consumption of the shaft. It was treated as a diabatic process without any thermal transmission. Sometimes, the air from the outlet piping was cooled first and then transferred to the cylinder through the backflow passage. In future, it is recommended to analyse how the cool backflow affects the performance of the Roots blower.

Disclosure statement

No potential conflict of interest was reported by the authors.

Funding

This study was supported by the National Natural Science Foundation of China (Research Project 51306136).

References

Arjeneh, M., Kovacevic, A., Rane, S., Manolis, M. J., & Stosic, N. (2015). Numerical and experimental investigation of pressure losses at suction of a twin screw compressor. Iop Conference Series: Materials Science and Engineering, 90(1), 1–10.

Broatch, A., Galindo, J., Navarro, R., Garcia-Tiscar, J., Daglish, A., & Sharma, R. K. (2015). Simulations and measurements of automotive turbocharger compressor whoosh noise. Engineering Applications of Computational Fluid Mechanics, 9(1), 12–20.

Burmistrov, A. A., Belyaev, L. V., Ossipov, P., Fomina, M., & Khannanov, R. N. (2001). Combined experimental and calculation study of conductance of Roots pump channels. Vacuum, 62(4), 331–335.

FLUENT Inc. (2013). Fluent theory guide. New York: Ansys Inc.

Hsieh, C.-F., & Deng, Y.-C. (2015). A design method for improving the flow characteristics of a multistage Roots pumps. Vacuum, 121(11), 217–222.

Hsieh, C.-F., & Hwang, Y.-W. (2007). Study on the high-sealing of Roots rotor with variable trochoid ratio. Journal of Mechanical Design, 129(12), 1278–1284.

Hsieh, C.-F., & Zhou, Q.-J. (2015). Fluid analysis of cylindrical and screw type Roots vacuum pumps. Vacuum, 121(11), 274–282.

Huang, Z. F., & Liu, Z. X. (2009). Numerical study of a positive displacement blower. Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science, 223(10), 2309–2316.

Hwang, Y.-W., & Hsieh, C.-F. (2006). Study on high volumetric efficiency of the Roots rotor profile with variable trochoid ratio. Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science, 220(9), 1375–1384.

Joshi, A. M., Blekhman, D. I., Felske, J. D., Lordi, J. A., & Mollen- dorf, J. C. (2006). Clearance analysis and leakage flow CFD model of a two-lobe multi-recompression heater. International Journal of Rotating Machinery, 2006, 1–10.

Kang, Y.-H., Vu, H.-H., & Hsu, C.-H. (2012). Factors impacting on performance of lobe pumps: A numerical evaluation. Journal of Mechanics, 28(02), 223–238.

Kovacevic, A., & Smith, I. K. (2002, January). The influence of rotor deflection upon screw compressor performance. Conference on screw type machines, VDI-Schrauben machinen. Dortmund, Germany.

Kovacevic, A., Stosic, N., & Smith, I. (2007). Screw compressors: Three dimensional computational fluid dynamics and solid fluid interaction. New York: Springer-Verlag Berlin Heidelberg.

Li, Y., Jia, K., Meng, Q. W., Shen, H., & Sang, X. H. (2013). Flow simulation of the effects of pressure angle to lobe pump rotor meshing characteristics. Iap Conference Series: Materials Science and Engineering, 52(3), 1–7.

Li, X. H., Zhang, S. J., Zhu, B. L., & Hu, Q. B. (2007, December). The study of the k-ε turbulence model for numerical simulation of centrifugal pump. International conference on computer-aided industrial design and conceptual design. Hangzhou, China.

Litvin, F. L., & Feng, P.-H. (1996). Computerized design and generation of cycloidal gearings. Mechanism and Machine Theory, 31(7), 891–911.

Mimmi, G., & Pennacchi, P. (1999). Analytical model of a particular type of positive displacement blower. Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science, 213(5), 517–526.

Ohtani, I., & Iwamoto, T. (1981). Reduction of noise in Roots blower. JSME International Journal Series B-Fluids and Thermal Engineering, 24(189), 547–554.

Peng, X. Y. (2000). Design improvement on involute profile of Roots blower rotors. Compressor Blower & Fan Technology, 3, 3–5. [Dictionary of the Chinese language]

Stosic, N., Smith, I. K., Kovacevic, A., & Mujic, E. (2011). Geometry of screw compressor rotors and their tools. Journal of Zhejiang University-Science A, 12(4), 310–326.
Sun, S. K., Zhao, B., Jia, X. H., & Peng, X. Y. (2017). Three-dimensional numerical simulation and experimental validation of flows in working chambers and inlet/outlet pockets of roots pump. *Vacuum*, 137, 195–204.

Tong, S.-H., & Yang, D. C. H. (2000). On the generation of new lobe pumps for higher pumping flowrate. *Mechanism and Machine Theory*, 35(7), 997–1012.

Tong, S.-H., & Yang, D. C. H. (2005). Rotor profiles synthesis for lobe pumps with given flow rate functions. *Journal of Mechanical Design*, 127(2), 287–294.

Valdes, L.-C., Barthod, B., & Perron, Y. L. (1999). Accurate prediction of internal leaks in stationary dry Roots vacuum pumps. *Vacuum*, 52(4), 451–459.

Wang, P.-Y., Fong, Z.-H., & Fang, H. S. (2002). Design constraints of five-arc Roots vacuum pumps. *Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science*, 216(2), 225–234.

Yang, D. C. H., & Tong, S.-H. (2002). The specific flowrate of deviation function based lobe pumps—derivation and analysis. *Mechanism and Machine Theory*, 37(10), 1025–1042.

Yao, L., Ye, Z., Dai, J. S., & Cai, H. (2005). Geometric analysis and tooth profiling of a three-lobe helical rotor of the Roots blower. *Journal of Materials Processing Technology*, 170(1), 259–267.