Three-dimensional numerical simulation and experimental validation of flows in working chambers of roots blowers with backflow design

S K Sun, Q Zhou, J Wen, X Y Peng*

School of Energy and Power Engineering, Xi’an Jiaotong University, No.28 West Xianning Road, Xi’an, Shaanxi, P.R.China, 710049.

*Email: xypeng@mail.xjtu.edu.cn

Abstract. The backflow design is one way to reduce the noise in the Roots blower. In this paper, the three-dimensional computational fluid dynamics (CFD) models were established to simulate the flows in the working chambers of the Roots blower with and without backflow design, using the dynamic mesh method. And then, the variations of pressure in the cylinder of the Roots blower with backflow design were tested in experiment to validate these CFD models. The result shows the CFD models were appropriate. The backflow could reduce the amplitude of the pressure pulsation in the outlet of the Roots blower more than 50%, but the volume efficiency also be reduced by 3.5%.

1. Introduction

The Roots blower is a positive displacement machine that is widely used in many fields, including mining, oil refining, the chemical industry or in chemical and environmental protection. The Roots blower has double rotors, with two or more lobes rotating in opposite directions in the cylinder. These double rotors are non-contacted and driven by a couple of synchronized gears. The rotors and the cylinder wall consist a working chamber which turn on and off periodically to transport fluid from the inlet to outlet. The pressure of the fluid does not rise in the transport process. The high pressure fluid in the outlet pocket flow back to compress the fluid in the working chamber at the moment the working chamber connected to the outlet pocket.

Many researchers have focused their attention on the profile design and meshing analysis of Roots blowers. Mimmi [1] optimized the parameters of the screw rotor based on a mathematical model of the
lobe profiles. Valdès [2] used the semi-empirical Knudsen–Dong law to predict the conductance of each clearance within a few percentage points for a wide range of pressures covering the transient flow regime. Tong and Yang [3,4,5] explored a new method of profile design by deriving a function to help choose the most efficient lobe parameters for high-sealing Roots blowers, according to the given flow rate functions. Burmistrov [6] proposed an angular coefficients method to calculate channel conductance in Roots blowers, based on the similarities between the laws of radiation and those of the diffusive reflection of molecules. Wang [7] designed a tooth profile consisting of five arcs, which obtained a better sealing performance due to its longer and narrower clearance. Yao [8] designed a novel three-lobe helical rotor for the Roots blower, with the cross section designed as a combination of concave arcs, convex arcs and cycloidal curves. The simulated cutter trajectory of the new rotor was then analyzed. Yao also devoted much effort to investigating the characteristics of the flows and leakages in the rotor’s working cycle. Hwang [9,10] proposed a new rotor profile with a variable trochoid ratio, and investigated how to achieve higher volumetric efficiency and higher sealing performance. Chiu [11] developed a new profile by using an elliptical roulette, and then compared the performances of the new and the traditional profiles.

With the development of computational fluid dynamics (CFD), growing numbers of researchers have focused on the numerical analysis of the Roots blower. Joshi [12] assumed that clearance between the impellers was almost inevitable during operation, and therefore built a static model to simulate the leakage through the clearance. Huang [13] used the k-ε model to simulate a three-lobe Roots blower, and compared the results of this simulation with those from semi-empirical formulas in terms of their non-uniformity of outlet flows. Li [14] investigated the effects of the rotor’s pressure angle on the flow characteristics by using a two-dimensional turbulence model. Hsieh [15,16] compared the differences between cylindrical and screw-type Roots blowers in terms of average outlet flow rates and instant outlet flow rates, based on a CFD case, namely BlowerLinx. Hsieh also analyzed the effects of the rotor phase angle in multi-stage Roots blowers with serial or parallel connections in terms of their flow rates and pulsation.

The Roots blower often working with big noise. The pressure pulsation is the major reasons of the aerodynamic noise, and backflow design is a good way to reduce the airflow pulsation. In this study, we established three-dimensional computational fluid dynamics (CFD) models of the Roots blower with and without backflow to analyze the flow character in the working circle. An experimental test rig was built with a novel Roots blower which the size identical to the simulated models. The pressures distribution in the working chamber and the inlet/outlet pockets were measured with five pressure sensors, and both the pressure and the flow rate results were used to verify the simulation result. Based on the CFD analytical and experimental investigations, the differences between the Roots with and without backflow design were discussed.

2. Numerical simulation

2.1 The structure of the Roots blower.
There are two parallel shafts rotating synchronously in opposite directions in a Roots blower, each of which is supported by two bearings, and one rotor and one gear are assembled on it. In a traditional Roots blower, the rotors were set in the middle of the shafts and the bearings were assembled on the two
side of the rotors. But in the Roots blower we used, the bearings were set at the two side of gears, as shown in Fig. 1. All of the bearings were set on one side of the rotors, so that only a front plate was needed on the free side of the rotors to cover the cylinder. The clearance in the cylinder could be easily measured by open the front plate. Furthermore, the pressure sensors could be conveniently installed on the front plate to measure the pressure in the cylinder. There are eight backflow holes at the cylinder wall, through which the higher pressure gas can improve the pressure in the working chamber at the position of 120° of the rotor’s angle, four on the upper working chamber and the other four on the downer working chamber. In this paper, the four horizontal backflow holes were closed by valves, so only four vertical backflow holes worked.

The Roots blower used in this study was a three-lobe blower. The diameter of the cylinder was 52.5 mm, and the length was 70 mm. The center distance of the two rotors was 70 mm, the profile of which consisted of six parts, as follows: two lines, three arcs and one involute [17]. The clearance between the two rotors was 0.14 mm, and that between the rotor and cylinder was 0.07 mm. The clearance between the rotor and the front plate was 0.11 mm, and that between the rotor and the back plate was 0.03 mm.

2.2 The geometry and grid of the Roots blower with backflow structure

In this study, the numerical simulation is the fluid dynamics analysis. So the fluid space was chosen as the computational domain. The clearances between the rotors and cylinder consist of two parts: circumferential and radial directions. The clearance of the radial directions was ignored because it is hard to achieve by dynamic mesh method. But the clearance between the rotors and cylinder were increased to 0.15mm to keep the total leakage no change. Onto the above-described geometry, inlet/outlet piping was added. The geometrical model of the Roots blower was established. The flow field consisted of the cylinder, inlet piping, inlet buffer area, outlet piping, outlet buffer area and backflow piping.

The grids of the three-dimensional Roots blower with backflow design was shown in Fig. 2. The inlet piping, inlet buffer area, outlet piping and outlet buffer area used hexahedral elements, and the cylinder used triangular prism elements, and the pre-flow piping used tetrahedral mesh. The total number of elements was 813221. The grids of the Roots blower without backflow was the same to the model with backflow except the interfaces between the backflow piping and cylinder were set to wall.
2.3 Governing equations and boundary conditions

In the numerical simulation based on the CFD software Fluent, the governing equations included continuity, momentum and energy conservation equations as follows [18]:

\[
\frac{\partial p}{\partial t} + \nabla \cdot (\rho \vec{v}) = 0 \tag{1}
\]

\[
\frac{\partial}{\partial t} (\rho \vec{v}) + \nabla \cdot (\rho \vec{v} \vec{v}) = -\nabla p + \nabla \cdot \vec{\tau} + \rho \vec{g} + \vec{F} \tag{2}
\]

\[
\frac{\partial}{\partial t} (\rho E) + \nabla \cdot (\vec{v}(\rho E + p)) = -\nabla \cdot \left( \sum h_j I_j \right) + S_h \tag{3}
\]

Where \( p \) was the static pressure, \( \vec{\tau} \) was the stress tensor, and \( \rho \vec{g} \) and \( \vec{F} \) were the gravitational body force and the external body force, respectively.

The realizable \( k-\varepsilon \) model was chosen for the turbulence equation because it had been validated for rotating shear flows and separated flows [18]. The near-wall treatment used the scalable wall function. The pressure-inlet and pressure-outlet boundary conditions were adopted. The gauge pressure of both inlet and outlet were set to 0 kPa. Although the outlet pressure was set to 0 kPa, the average pressure in the outlet pocket of the Roots blower was 50 kPa. It is because there was a narrow pipe which throttle the fluid to increase the pressure before the fluid reached the outlet buffer area. The fluid was air with properties as idea-gas. The connections between adjacent parts used interface method to transfer data.

The working process of a Roots blower is a transient problem, so it is necessary to bring in the dynamic mesh. The local-remeshing and the spring-based smoothing dynamic mesh methods were adapted in the cylinder areas. A UDF (user defining function) was programed to drive the rotors rotating as rigid bodies.

The space conservation equations of the dynamic meshes is an integral form of the conservation equation for a general scalar \( \Phi \) on an arbitrary control volume \( V \), whose boundary is moving can be written as [18]
\[
\frac{d}{dt} \int_V \rho \phi dV + \int_V \rho (\vec{u} - \vec{u}_g) \cdot d\vec{A} = \int_V \nabla \phi \cdot d\vec{A} + \int_V S_{\phi} dV \quad (4)
\]

where \( \rho \) is the fluid density, \( \vec{u} \) is the flow velocity vector, \( \vec{u}_g \) is the mesh velocity of the moving mesh, \( \Gamma \) is the diffusion coefficient, \( S_{\phi} \) is the source term of \( \phi \), and \( \partial V \) is used to represent the boundary of the control volume \( V \).

The time step was determined by \( \Delta t = \frac{\text{CFL} \Delta x}{\lambda_{\text{max}}} \), where CFL was the Courant number, \( \Delta x \) was the grid distance, and \( \lambda_{\text{max}} \) was the maximum velocity scale. Therefore, the time step was set up as \( 1 \times 10^{-5} \) s.

3. The experiment on the Roots blower

The equipment for the experimental investigation of the Roots blower was shown in Fig. 3. This equipment is similar to the simulation model. It consisted of a motor, a Roots blower, pressure gauges, inlet/outlet piping, valves, a flow meter, pressure sensors, backflow piping and an acquisition system. The pressure gauges and valves could measure and control the inlet and outlet pressure of the Roots blower. The pipe behind the outlet valves was connected to a flow meter. There were eight backflow pipes connect to the outlet piping, each of them contain a valve to control whether this pipe is on work. In order to accord to the simulation models with backflow, four of the valves were closed. When the data of the Roots without backflow, the eight valves were all closed.

In this research, the purpose of the experiment was to verify whether the CFD simulation could provide an enough accuracy result for the Roots blower with backflow. So the pressure distribution in working chamber and inlet/outlet pocket was need to compare with the simulation one. Therefore five sensor were installed on the front plate of the Roots blower. Fig. 4(a) shows the exactly position of these
sensors. Sensor 1 was to measure the pressure in the inlet pocket, sensor 2, 3 and 4 were to measure the pressure in the working chamber, sensor 5 was to measure the pressure in the outlet pocket. All the sensor were installed around the upper rotor, because the double rotors were fitted nearly symmetrically except a difference of 60° in the phase angle.

The rotor rotated 300° during a working circle as shown in Fig. 4(b-g). The working chamber of Roots blower closed at 120° and opened at 180°. The available test angle of sensors were shown in Table. 1. So the pressure of the working chamber could be measured at any time.

**Table.1** The available test angle of the pressure sensors.

| Sensor number | angle            |
|---------------|------------------|
| Sensor 1      | 0° ~ 60°         |
| Sensor 2      | 60° ~ 120°       |
| Sensor 3      | 120° ~ 180°      |
| Sensor 4      | 180° ~ 240°      |
| Sensor 5      | 240° ~ 300°      |

![Fig. 4](image)

**Fig. 4** The positions and the available test angles of the pressure sensors.

With the experimental equipment introduced above, the dynamic pressure in the working chamber, inlet and outlet pocket of the Roots blower could be acquired. The experimental data were obtained under the following conditions: The motor was rotating at 2400 rpm. The valve in the inlet piping was fully open, and the valve in the outlet piping was adjusted to ensure that the outlet pressure was 50 kPa. The measuring range of the pressure sensors was 0~3.5 bar (absolute pressure), and the sample rate of the sensors was 10 kHz.

4. The results and discussions

4.1 Comparison of experimental and simulation results for the flowrate

The numerical simulation performed in this study involved a transient problem with an initial value. Before the simulation achieved a periodic steady state, it was necessary to calculate a large number of time steps. One of the choices for determining if the simulation was steady was to decide whether the
difference in the average mass flow between the inlet and outlet during a working cycle could be ignored. Generally, the simulation could be considered steady if the difference was lower than 5%. Table. 2 shows the average mass flow rate for the two cases in the third working cycle, from which we could determine that the simulations had been periodically steady while the calculations were run to the third period.

Both the mass flow rates of the simulation and the experiments were shown in Table. 2. It shows the simulation results was 6.5% lesser than the experiments one for Roots without backflow and 4.0% lessor for Roots with backflow. Both the simulations results were lesser than the experiments results, it is because it hard to achieve accurate measurements of the size of clearance. Consider the differences were small, the results of the simulation were accredited. The Table. 2 also shows the mass flow rates of the Roots blower with backflow was lesser than the one without backflow. It is because the pressure of the working chamber of the Roots blower with backflow was higher than the blower without backflow. The higher pressure between the working chamber and the inlet lead to more leakage, which reduce the mass flow rate on the outlet. The mass flow rate of the Roots blower with backflow was about 3.5% lesser than the one without backflow in this study.

| Table.2 | The average mass flow rates of the simulations and the experiment. |
|---------|---------------------------------------------------------------|
|         | Experiment(kg/s) | CFD simulation(kg/s) |
|         | inlet          | outlet              |
| Roots without backflow | 0.01475 | 0.01554 | 0.01579 |
| Roots with backflow     | 0.01423 | 0.01456 | 0.01504 |

4.2 Comparison of experimental and simulation results for the variations in pressure

Fig. 5 The variations in pressure of the rotor’s angle in a working cycle.
According to the available test angles of the pressure sensors as shown in Table.2, the variations of pressure
in the working chamber, with the rotor rotating angles, are shown in Fig. 5. The results between the experiments and the simulation for both the Roots blower with and without backflow agreed with each other well. It is said that the CFD simulation results were believable.

Fig. 5 shown the variations in pressure for the Roots with backflow were almost the same to the Roots without backflow from 0° to 120°, because the working chamber was connected to the inlet pocket and disconnect to the backflow piping in this time, so the impact of the backflow could hardly show up. But since 120°, the pressure variation is difference. For Roots blower without backflow, from 120° to 180°, the working chamber was closed, the higher pressure air leak from outlet pocket through the clearance between the rotor and the cylinder to the working chamber, which made the pressure in the working chamber increased slowly. Since 180°, the working chamber connected to the outlet pocket, the air in the outlet pocket flow back to the working chamber because the pressure in the working chamber was lower than the outlet. The direction of the flow back was opposite to the rotor rotating, which caused impact and pulsation of the air. The amplitude of pressure pulsation was about 25kPa. For Roots blower with backflow, from 120° to 180°, the working chamber was disconnect to the inlet pocket and connect to the backflow piping, the higher pressure air flow into the working chamber and increased the pressure in the chamber. When the working chamber connect to the outlet pocket, the pressure in the working chamber was close to the outlet pocket. So the pressure pulsation in the outlet pocket was smaller than Roots blower without backflow, it is only 10kPa. Both the simulation and the experiment shown the backflow can reduce the amplitude of pressure pulsation of the Roots blower.

4.3 The pressure field

![Image of pressure field with various angles](image)

**Fig. 6.** The comparison of the Roots blower with and without backflow on pressure field.
The symmetry plane was chosen as the observe plane to study the difference between the Roots blower with and without backflow on the pressure field. Fig. 6 shows the pressure field of these two cases.

In the Roots blower without backflow, the pressure difference between the working chamber and outlet pocket was about 40kPa at 180° (Fig. 6(a)). When the working chamber connected to the outlet pocket, the large pressure difference run the air in the outlet pocket flow back to the working chamber which reverse the air flow in the working chamber and lead to air impact at 190° (Fig. 6(b)). The air in the working chamber was compressed by the flow back air impact, and the pressure in the outlet pocket reduced because the air flow back to the working chamber at 200° (Fig. 6(c)), so a new pressure difference was generated. The direction of the air flow was changed again. But in the Roots blower with backflow, the pressure difference was much less than Roots without backflow. It is benefit from the backflow piping help the pressure in the working chamber close to the outlet pocket, which reduce the pressure impact and the amplitude of the pressure pulsation.

5. Conclusions
Based on the CFD analytical and experimental investigations presented in this study, the following conclusions may be drawn:

(1) The backflow structure can help reduce the pressure pulsation in the outlet of the Roots blower. The amplitude can be reduced more than 50%.

(2) The backflow structure could reduce the mass flow rate of the Roots blower, because the higher pressure difference between the working chamber and the inlet pocket generated more leakage. The volume efficiency could be reduced about 3.5%.

(3) The main reason of the pressure pulsation in Roots blow was the air impact caused the flow back from outlet pocket to the working chamber.

This study’s procedure for the analysis of the Roots blower with backflow provides a good basis for further consideration of leakage and noise reduction.

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

References
[1] Mimmi G, Pennacchi P, Mimmi G. 1999, Analytical model of a particular type of positive displacement blower. ARCHIVE Proceedings of the Institution of Mechanical Engineers Part C Journal of Mechanical Engineering Science, 213(5): pp.517-526.
[2] Valdès, L.C, Barthod. B, Perron. Y.L, 1999, Accurate prediction of internal leaks in stationary dry Roots vacuum blowers. Vacuum, 52 (4), pp.451–459.
[3] Tong S H, Yang D C H. 2000, On the generation of new lobe blowers for higher blowing flowrate. Mechanism & Machine Theory, 35(35): pp. 997-1012.
[4] Yang D C H, Tong S H. 2002, The specific flowrate of deviation function based lobe blowers - derivation and analysis. Mechanism & Machine Theory, 37(10): pp. 1025-1042.
[5] Tong S H, Yang D C H. 2005, Rotor profiles synthesis for lobe blowers with given flow rate
functions. Journal of Mechanical Design, 127(2): pp. 287-294.

[6] Burmistrov. A, Belyaev. L, Ossipov. P, Fomina. M, Khannanov. R, 2001, Combined experimental and calculation study of conductance of Roots blower channels. Vacuum, 62 (4) pp.331–335

[7] Wang P Y, Fong Z H, Fang H S. 2002, Design constraints of five-arc Roots vacuum blowers. Proceedings of the Institution of Mechanical Engineers Part C, 216(2): pp. 225-234.

[8] Yao L, Ye Z, Dai J S, 2005, et al. Geometric analysis and tooth profiling of a three-lobe helical rotor of the Roots blower. Journal of Materials Processing Technology, 170(1–2): pp. 259-267.

[9] 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): pp. 1375-1384.

[10] 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): pp. 1278-1284.

[11] Hsieh C F. 2015, A new curve for application to the rotor profile of rotary lobe blowers. Mechanism & Machine Theory, 87: pp. 70-81.

[12] Joshi A M, Blekhman D I, Felske J D, et al. 2006, Clearance analysis and leakage flow CFD model of a two-Lobe multi-recompression heater. International Journal of Rotating Machinery, (1023-621X): pp. 10.

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

[14] Li Y B, Jia K, Meng Q W, et al. 2013, Flow simulation of the effects of pressure angle to lobe blower rotor meshing characteristics. 6th International Conference on Blowers and Fans with Compressors and Wind Turbines, Materials Science and Engineering, 52(3):032022.

[15] Hsieh C F, Deng Y C. 2015, A design method for improving the flow characteristics of a multistage Roots blowers. Vacuum, 121: pp. 217-222.

[16] Hsieh C F, Zhou Q J. 2015, Fluid analysis of cylindrical and screw type Roots vacuum blowers. Vacuum, 121: pp. 274-282.

[17] XY Peng et al. 2000, Design improvement on involute profile of roots blower Rotors. Compressor blower & fan technology 2000(3): 3–5. (In Chinese)

[18] FLUENT 15.0, 2013 User help.