CFD Analysis of Leakage Flow in Radial Tip Gap of Roots Blower

Neeraj Bikramaditya, Sham Rane, Ahmed Kovacevic, and Brijeshkumar Patel

Abstract A Roots Blower is rotary positive displacement machine, commonly used for low pressure applications [3]. However, the gaps between the rotors and the housing are the main source of volumetric inefficiency and are required to be minimised. This has limits due to thermal expansion of compressor elements. Improvements can also be done by minimising leakage flows using different configurations of rotor tip profiles for which careful analysis is required. An optical Roots Blower from Howden is being investigated using experimental and numerical tools for the effects of heat transfer and tip geometry on leakage of gas. To closely study the leakage through the clearance gaps, a 2D simplification (Fig. 1b) of this 3D model is proposed in this paper. A local flow is evaluated in steady and transient state conditions using only through the tip leakage gap between the rotor and the housing on one rotor lobe. Using data from PIV measurements, the base tip design on the rotor profile is analysed and used for validation of the 2D model. Following this, variants of the tip-shape, namely equal-cavity and unequal-cavity tip profiles with alterations, have been numerically evaluated. These results will help in implementation of such a tip profile design in conventional oil free twin screw compressors to meet demands of efficiency improvements.

Keywords CFD · PIV · Roots blower · Leakage · Clearance · PR (pressure ratio)

1 Introduction

Roots blower is a Rotary PDM used for low pressure applications. It is also known as straight lobe compressor. This oil free air delivery machine is useful for the industries where contaminations plays an important role such as FMCG, Chemical, Pharmaceutical, textile etc. The Roots Blower has oppositely rotated and non-contacting pair of Rotors enclosed within a casing. One of the rotors is known as main/male rotor and other as gate/female rotor. There are three types of gaps in the blower, namely

N. Bikramaditya (✉) · S. Rane · A. Kovacevic · B. Patel
City, University of London, London EC1V 0HB, UK
e-mail: Neeraj.Bikramaditya@city.ac.uk

M. Read et al. (eds.), *13th International Conference on Compressors and Their Systems*, Springer Proceedings in Energy, https://doi.org/10.1007/978-3-031-42663-6_4

inter-lobe or rotor-to-rotor gap, the tip or rotor-to-casing gap and the axial gap. They are separated by a precisely engineered gaps through which certain amount of air escape as losses. As shown in Fig. 1a, the Roots blower uses two straight-shaped lobe impellers mounted on parallel shafts. When the lobe passes over the blower inlet, a finite volume of air is trapped and is carried around the chamber by the lobes. The air is then discharged at the blower outlet. As the lobes continue to rotate, the pressure increases in the reservoir beyond the blower outlet. Thus, the pressure difference between discharge and suction causes air to flow back from the reservoir to the low-pressure regions through these clearances. To make flowing air be oil free and flow without lubrication, these clearances between the rotors (Lobe) and between the rotors and the casing (Tip and End plate) are kept.

Researchers have been trying to study the physics of leakage flow to understand the flow phenomena in the clearance gaps. The gap management plays crucial role for better reliability and efficiency of the machine. Brijesh et.al [7] have developed and outlined the PIV method to capture important flow physics using the velocity and temperature field. Vimmr [1] have presented a numerical simulation of the leakage flow between the moving rotor and housing of a screw compressor. In their 2D moving mesh for analysis, they observed that rotor speed has negligible effect on velocity field at given pressure ratio. The fourth author of this paper has conducted the PIV test on roots blower for his PhD thesis, has found out that there is change in the velocity field in the leakage gap and in the downstream. In the present study, the goal was to develop a simplified model of the Roots blower which could validate the PIV results and efficiently be used for the modelling of different shapes of the Rotor tip.

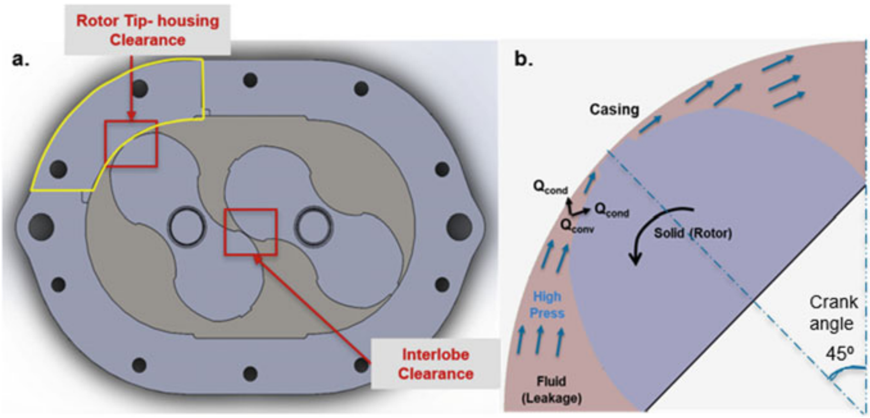

Fig. 1 **a** Section view of roots blower clearances, **b** fluid flow in clearance roots blower

2 Computational Fluid Dynamics

Fluid flow is governed by three fundamental conservation laws of mass, momentum and energy. CFD employs numerical methods and algorithms to solve mathematical models which describe fluid flow using governing equations to a large set of algebraic equations. In the era of high computational capability, CFD has tremendous the ability to produce accurate solution for complex and realistic geometries. ANSYS Fluent commercial CFD code is used which is based on Finite Volume Method using conservation laws of fluids. More details can be found in ANSYS-FLUENT Theory guide [2].

2.1 Conservation Laws of Fluids

Mass Conservation Equation. The mass conservation law states that the net mass crossing the boundary of a control volume must be balanced by an accumulation or depletion of mass in that control volume. For compressible flow, the mass can increase or decrease within the control volume. Mass conservation equation or equation of continuity is mathematically defined as:

$$\frac{\partial \rho}{\partial t} + \frac{\partial \left(\rho v_j\right)}{\partial x_j} = 0 \tag{1}$$

Momentum Conservation Equation. The momentum conservation equations are derived from the second Newton's law of motion. It states that the sum of the forces acting on a fluid particle is equal to the mass of the element multiplied by its acceleration. The formulation below is a 3D transient formulation of the Naviers-Stokes equations for compressible flow in Eulerian frame of reference:

$$\frac{\partial (\rho v_i)}{\partial t} + \frac{\partial \left(\rho v_j v_i\right)}{\partial x_j} = \frac{\partial}{\partial x_j}\left(-p\delta_{ji} + \Sigma_{ji}\right) + \rho f_i \tag{2}$$

Energy Conservation Equation. The energy conservation equation is derived from the first law of thermodynamics which states that energy can't be produced or destroyed, just converted from one form to another. The change in energy over time is equal to the sum of the work done and the thermal energy generated:

$$\frac{\partial (\rho C_v T)}{\partial t} + \frac{\partial \left(\rho C_v v_j T\right)}{\partial x_j} = -p\frac{\partial v_j}{\partial x_j} + \Sigma_{ji}\frac{\partial v_i}{d x_j} + \frac{\partial}{\partial x_j}\left(\lambda \frac{\partial T}{\partial x_j}\right) + \phi v \tag{3}$$

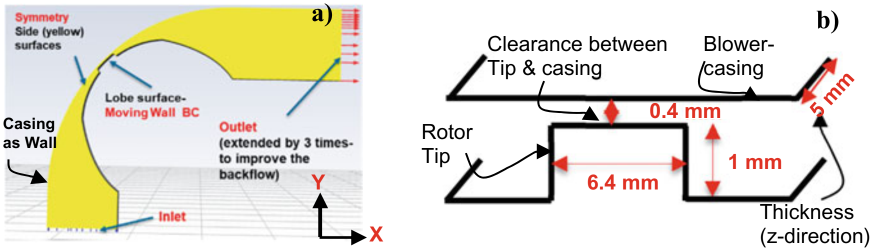

Fig. 2 **a** Computational domain (simplified 2D). **b** Schematic L/O of leakage region

The basic variables used in Eqs. (1), (2) and (3) ρ, v, p, f, C_v, $T and \lambda$ are density, velocity, pressure, specific heat capacity, temperature and thermal conductivity respectively. For more details about notations, ANSYS [2] can be referred.

3 Computational Analysis of Roots Blower

The ANSYS Fluent Commercial code is used for the simulation of the modified 2D of 3D Roots blower. The geometry is created using ANSYS Workbench and Mesh is created using ANSYS Mesh.

3.1 Computational Domain of Simplified Roots Blower

Figure 2a is the simplified domain considered for the validation of the CFD set up. The modelling consists of defining input conditions and related boundary conditions, turbulence models, solution methods with both static and transient mode calculations.

The Pressure inlet and pressure outlet boundary conditions is used for inlet and outlet respectively. Since the 2D model with thickness is considered, both side surfaces (yellow) are used as periodic BC. Figure 2b shows the dimensions of the modelling.

3.2 Simulation Set-Up

Air has been used as a working medium. It is considered as a perfect gas which means that its density depends on temperature and pressure. It has density of ideal gas, specific heat of 1006.43 J/(kgK), thermal conductivity of 0.0242 W/(mK) and viscosity of 1.7894e − 05 kg/ms). Table 1. is the summary of the simulation cases considered same as PIV test.

Table 1 Input conditions from PIV test

Items	PR 1.6		PR 1.4		PR 1.2	
RPM	Pin/Pout (kPa)	Tin/Tout (K)	Pin/Pout (kPa)	Tin/Tout (K)	Pin/Pout (kPa)	Tin/Tout (K)
2000	161.2/ 100.8	438/390	143/102	364.3/304.9	121.6/ 100.8	330.8/303.1
1800		418/311		333.6/306.2		330.1/302.5
1500				380.1/309.6		332.1/302.1
1000						329.1/302.1

Table 2 Solver setting

Items	Specification	Items	Specification
Solver	Pressure based	Spatial discretization	2nd order upwind
Turbulence	K-ω SST, K-ε, LES	Turbulence numeric	2nd order upwind
Fluid medium	Air	Gradient	Green-Gauss node
P–V coupling	Coupled	Flux-type	Rhie-chow: mom based
Transient	1st order implicit	Time-step size	0.001(s)

The simulation settings are shown in Table 2. The turbulence was modelled with the Shear Stress Transport (SST) k-ω model, K-ε and LES. The k-ω SST turbulence model is selected mainly unless and otherwise stated differently in present calculations.

The flow was assumed to be subsonic below an overall pressure ratio of 1.9 and sonic above it [3]. The current set up is considered after careful consideration of the transient setup of Roots blower by Sun. et.al [6]. The domain and solvers are different from Sun et. al.

First the steady state calculation performed and after few iterations, transient simulations was adapted with Flow Courant number 20. This courant number is applied after investigating the convergence behavior. The Fluent default under-relaxation for body-force, k, omega, density and turb-viscosity is used.

4 Results and Discussion

Roots blower geometry and simulation set-up were presented in the previous sections. In the first section, the numerical results will be validated with PIV data and in the second section, the results will be compared for the different rotor tip design concepts and their effect. The Physical phenomena and the related analysis are presented.

4.1 Validation

For the validation of the current CFD set up, the absolute maximum velocity at the exit of the tip and the averaged velocity profile through leakage (casing to rotor) is considered. The Leakage gap is maintained at the 400 μm. The comparison conditions are made at PR 1.6, 1.4 and 1.2 with RPM 2000, 1800, 1500, 1000 and 0.

In order to validate the current CFD model, there were three basic turbulence modelling approach investigated at all Pressure ratio (PR) 1.6, 1.4 and 1.2, and K-omega and LES was chosen at PR1.4. The PIV results shows that at one fixed Pressure Ratio, different Rotor speeds (RPM) show variation in the velocity profiles between rotor tip and casing (i.e., Leakage gap). This PIV result pattern is not observed in CFD results. CFD has shown almost no variation of the velocity profile for all the Rotational speed of the Rotor.

At PR1.6, PIV showed higher velocity in leakage region for RPM1800 and RPM0 than RPM2000. CFD has shown (Figs. 3 and 4) a negligible variation in averaged velocity profiles for all the three Turbulence models (RANS (k-ω SST, K-ε) and LES) (Fig. 3). Although, there was some change observed for K-Epsilon but cannot be validated as an improvement because this pattern was not observed at other PRs.

For the PR1.4, PIV resulted in the similar trend as PR1.6 (Figs. 5 and 6). Averaged velocity profiles for the RPM1800 and RPM0 is higher than that of RPM2000 and RPM1500, but this flow characteristics is not visible in CFD for all the turbulence models (Fig. 5).

Also, at the PR1.2, RPM1800, RPM1000 and RPM0 are higher in vel. magnitude than RPM2000 and RPM1500 (Figs. 7 and 8). CFD has not depicted this vel. profile for any of the turbulence model chosen (Fig. 7).

There is a possibility that 2D-model and non-rotating mesh are combinedly unable to track the physics of the leakage flow and downstream flow field of tip. The mesh rotation usually impose fluctuation in the flow field through their complete rotation cycle (Figs. 3, 4, 5, 6, 7 and 8).

At the exit of the rotor tip, the maximum velocities were investigated to compare them with the PIV results (Fig. 9). Exit velocity indicates the speed at which the flow is exiting as leak and tend to predict the volume flow of leak.

The absolute max. velocity has also shown the better agreement for the RPM2000 at PR1.6. For all other cases, discrepancies exist within the range of 20%. At this particular case of PR1.6 and RPM2000, the averaged velocity profile through leakage and the abs. max. velocity at the tip exit are in closer agreement with PIV as shown in Fig. 10.

4.2 Tip Design Concept Study

Considering the case of PR1.6 and RPM2000, the Roots blower tip concepts will be investigated in this paper. The mass flow rate ($\dot{m} = \bar{U}*A$) on the section plane at

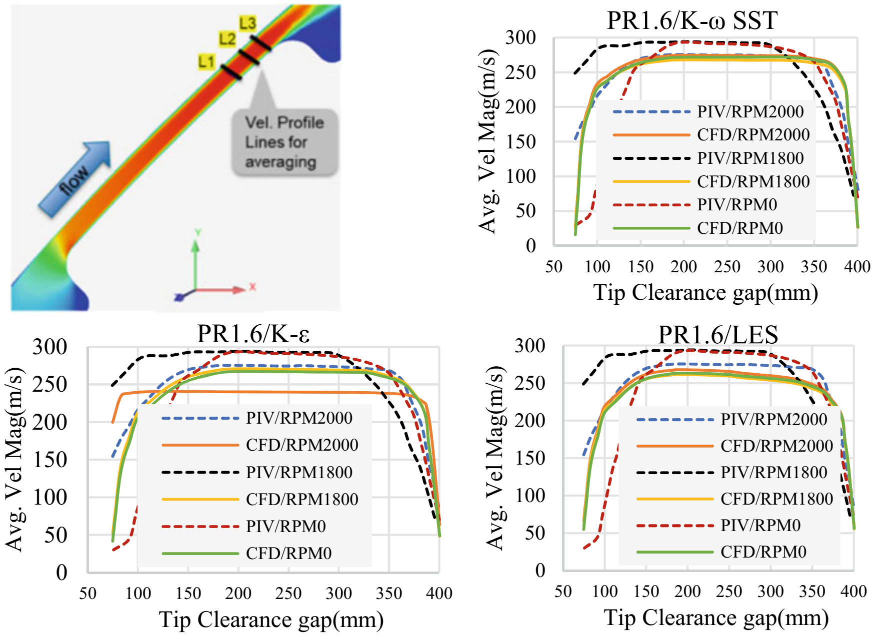

Fig. 3 **PR1.6**/velocity profiles comparisons K-ω SST versus K-ε and LES

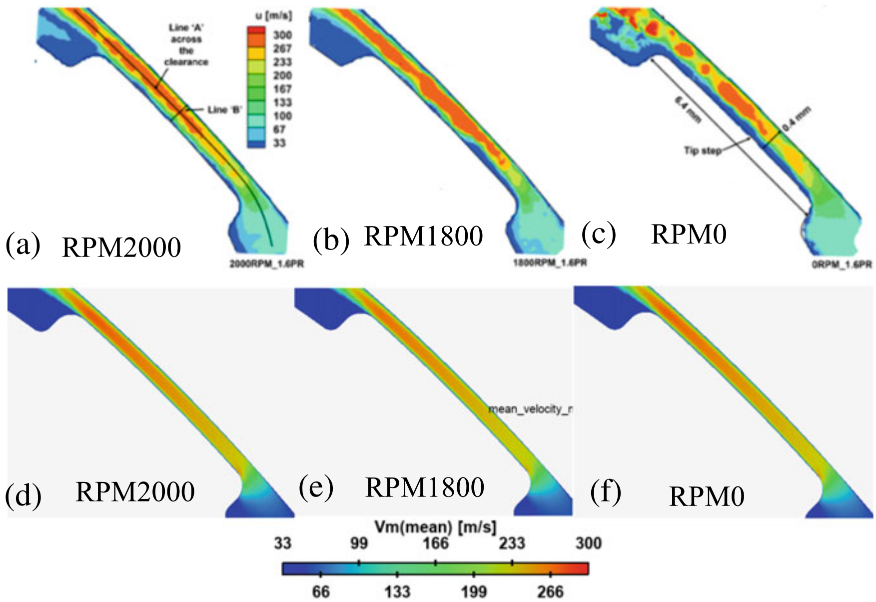

Fig. 4 **PR1.6**/vel. contours comparison b/w PIV (**a**–**c**) and CFD k-ω SST model (**d**–**f**)

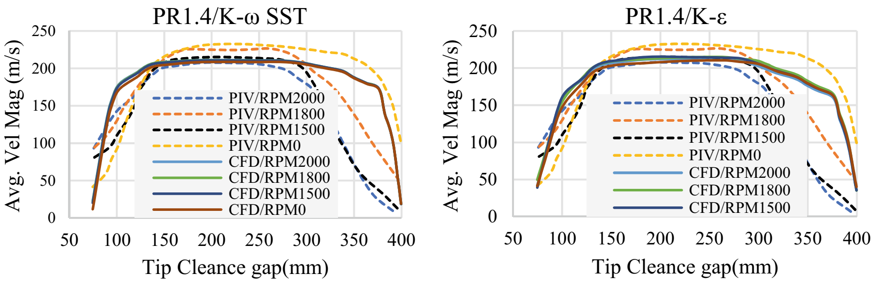

Fig. 5 **PR1.4**/velocity Profiles comparisons K-ω SST versus K-ε

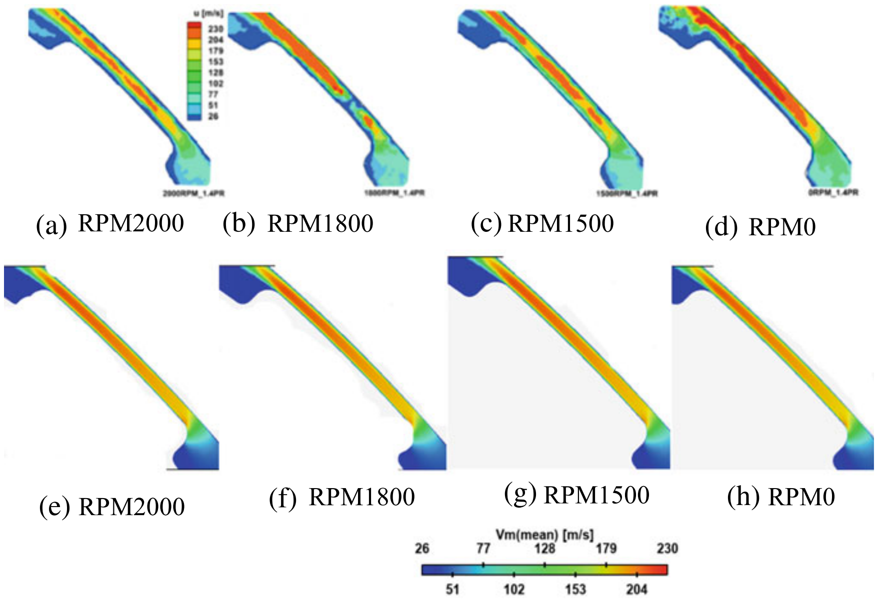

Fig. 6 **PR1.4**/vel. contours comparison b/w PIV (**a–d**) and CFD k-omega SST (**e–h**)

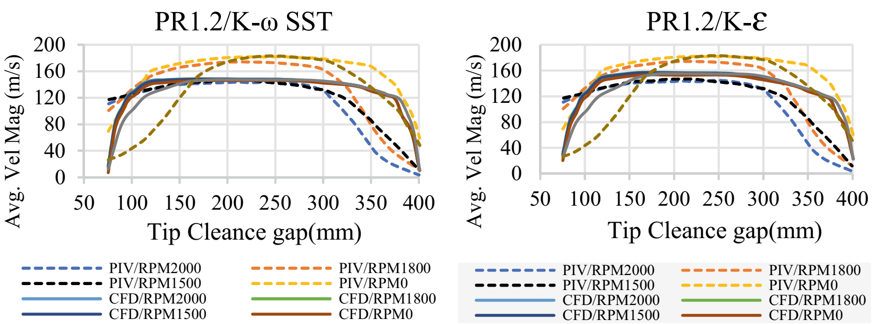

Fig. 7 **PR1.2**/velocity profiles comparisons K-ω SST versus K-ε

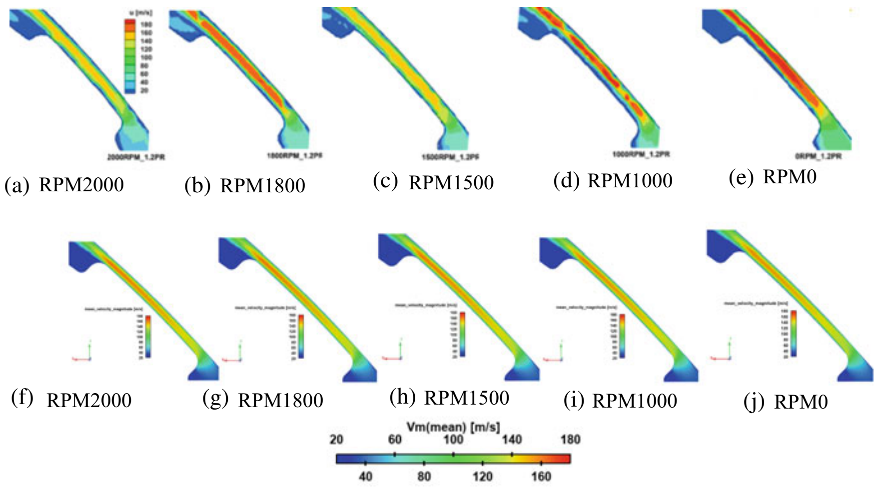

Fig. 8 **PR1.2**/vel. contours comparison b/w PIV (**a**–**e**) and CFD k-omega SST (**f**–**j**)

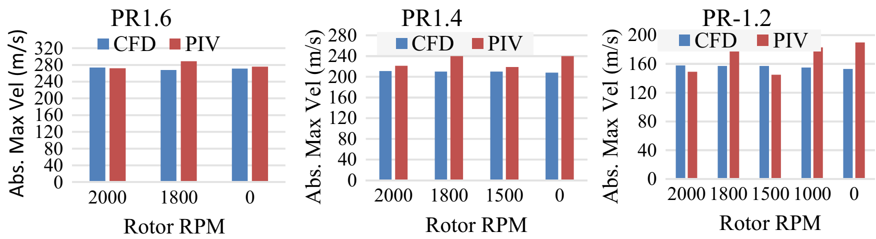

Fig. 9 Comparison of absolute max. vel at the tip exit

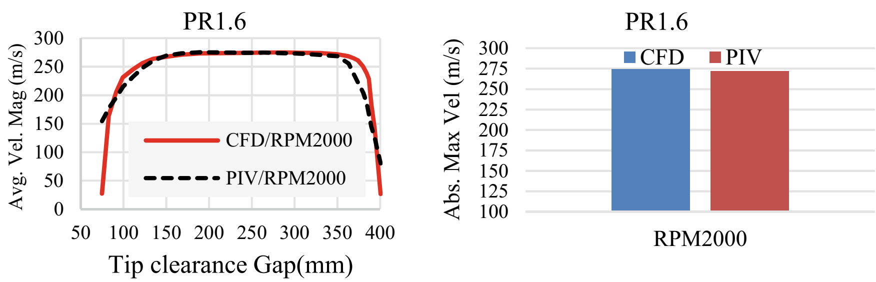

Fig. 10 Comparison of averaged vel. profile through leakage and absolute max. vel at the tip exit

the exit of the rotor is considered as a main evaluation criterion. The corresponding results will be discussed and more efficient design will further be considered for the future designs. Figure 11 is current combination of tip concepts.

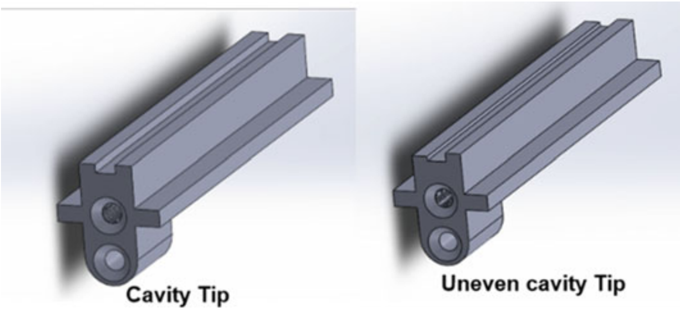

Fig. 11 Tip and cavity concepts

4.3 *Rotor Tip Design Study*

Three concepts of Tip are being analyzed using Even-cavity and Uneven-cavity. Figure 12(a) is Base-type, Fig. 12(b) is (b)-type even-cavity tip, Fig. 12(c) is (c)-type uneven-cavity tip with high tip at inlet side and Fig. 12(d) is (d)-type uneven-cavity tip with high at exit side (Fig. 12).

The contour comparison (Fig. 13) and mass flow rate comparison (Fig. 14(a) and (b) results show that (b)-type concept (Q1-(a)-type) has least leakage (improved by 21%) in the gap compared to other concepts (Fig. 14(b)). The cavity between the tip is working as the flow reduction in the downstream as it creates the vortices. Also, the next best tip concept is (d)-type (Q3-(c)-type) which has decreased the leakage 15% compared to (a) Base type (Q (Base Cavity)). The sudden restriction of flow after the bigger entrance of flow was able to reduce the flow at tip exit. The (c)-type (Q2-(b)-type) concept has increased the flow speed at entrance and finds wider passage at the exit tip to the downstream. The mass flow rate comparison graph clearly indicated the improved result and which will be helpful in deciding future shapes (Fig. 14(b)).

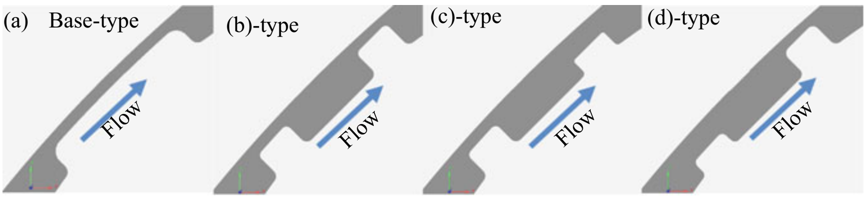

Fig. 12 Computational analysis domains of tip concepts (2D)

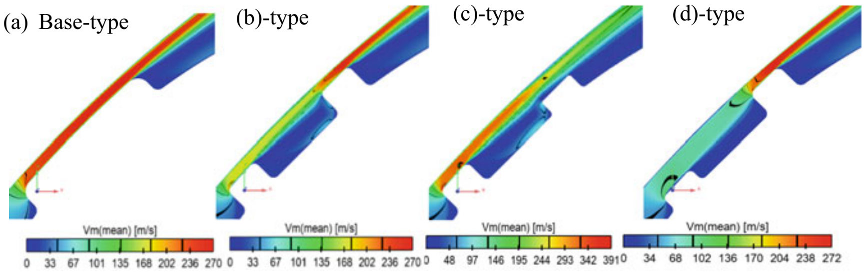

Fig. 13 Comparison of mean velocity magnitude contour through the tip gaps

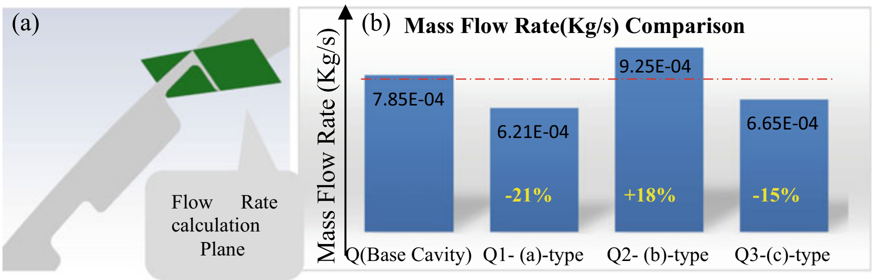

Fig. 14 **a** Location of mass flow rate measurement. **b** Result comparison between cavity tips

5 Conclusion and Future Scope

The presented simplified 2D model of Roots blower is aimed to develop a reliable CFD setup to study complex leakage flows thoroughly. Several cases were used for validation with PIV results and analyzed. Even though the average velocity profiles for most cases are not in very good agreement, Case with PR1.6 and RPM2000 is promising. Using this case, the cavity tip concepts were analyzed, and suitable tip to improve the leakage was found. As a future work, present CFD model will be improved to mimic the PIV test set-up using improved meshing techniques such as dynamics mesh and rotating mesh. Also, CHT [4, 5] with improved mesh model will be applied to make simulation more realistic.

Acknowledgements Funding for this research was received from Royal Academy of Engineering, UK and Howden, UK towards the project Smart Efficient Compression: Reliability and Energy Targets (SECRET).

References

1. J. Vimmr, Mathematical modelling of compressible viscous fluid flow in a male rotor-housing gap of screw machines. Proc. Appl. Math. Mech. **4**(1), 454–455 (2004)
2. ANSYS, Fluent theory guide
3. S. Mcdougald, B.W. Imrie, B.N. Cole, *An Investigation of the Volumetric Efficiency of a Roots Blower*. Purdue E-pubs (1974)
4. M. Matuzović, S. Rane, B. Patel, A. Kovačević, Ž. Tuković, Analysis of conjugate heat transfer in a roots blower and validation with infrared thermography. Int. J. Thermofluids, 100234 (2022)
5. S. Rane, A. Kovačević, N. Stošić, I.K. Smith, Bi-directional system coupling for conjugate heat transfer and variable leakage Gap CFD analysis of twin-screw compressors. IOP Conf. Ser. Mater. Sci. Eng. **1180**(1), 12001 (2021)
6. S. Sun, A. Kovacevic, C. Bruecker, A. Leto, G. Singh, M. Ghavami, Numerical and experimental analysis of transient flow in roots blower. **4**(1), 3 (2020)
7. B. Patel, A. Kovacevic, A. Krupa, On measuring velocity and temperature in leakage flows of oil free rotary positive displacement machines, in *New Technologies, Development and Application IV* (Springer International Publishing, Cham, 2021), pp. 763–773