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

Neeraj Bikramaditya, Sham Rane, Ahmed Kovacevic and Brijeshkumar Patel

City, University of London, London EC1V 0HB, UK
Neeraj.Bikramaditya@city.ac.uk

Abstract. A Roots Blower is rotary positive displacement machine, commonly used for low pressure applications. 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 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, two variants of the tip, namely equal-cavity and unequal-cavity tip profiles, 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 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 Figure 1 (a), 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.

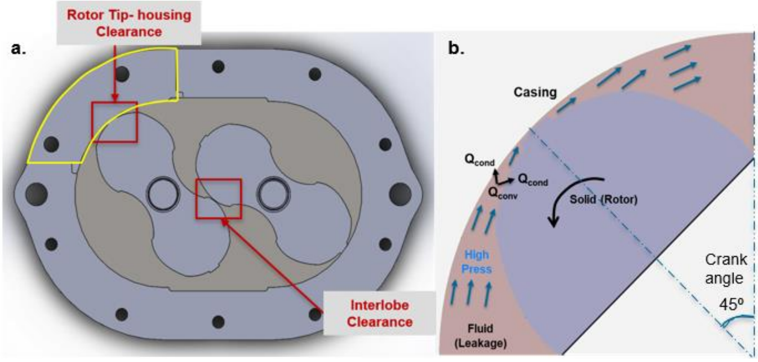


Fig. 1. (a) Section view of Roots Blower clearances (b) Fluid flow in clearance Roots Blower.

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 [1].

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:

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

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:

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

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:

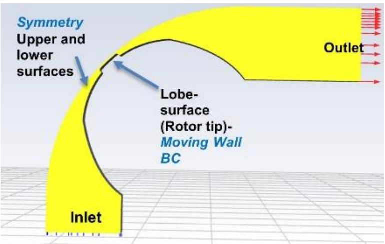
$$\frac{\partial(\rho C_v T)}{\partial t} + \frac{\partial(\rho C_v v_j T)}{\partial x_j} = -p\frac{\partial v_j}{\partial x_j} + \Sigma_{ji}\frac{\partial v_i}{d x_j} + \frac{\partial}{\partial x_j}\left(\lambda\frac{\partial T}{\partial x_j}\right) + \phi v \tag{3}$$

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

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



Region	BC
Inlet	Press Inlet/Tin
Outlet	Press Outlet/Tout
Upper & Lower face	Symmetry

Fig 2. Computational Domain (Simplified 2D model of 3D Roots Blower)

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.

Table 1. Fluid Properties.

Items	Fluid (Air) specification
Density [kg/m3]	Ideal-gas
Specific Heat [J/(kgK)]	1006.43
Thermal Conductivity[W/(mK)]	0.0242
Viscosity[kg/m-s)]	1.7894e-05

Table 2. 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

The simulation settings are shown in Table 3. 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.

Table 3. 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

The flow was assumed to be subsonic below an overall pressure ratio of 1.9 and sonic above it [2].

First the steady state calculation performed and after few iterations, transient simulations was adapted with Flow Courant number 20. At this particular Courant number, flow was varied with the momentum and pressure under-relaxation to check the stability of the calculation. The under-relaxation for body-force, k, omega, density and turb-viscosity considered are 0.6, 0.4, 0.4, 0.6 and 0.5 respectively.

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 its effect. The Physical phenomena and the related analysis will be 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 400µm. 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 a negligible variation in averaged velocity profiles for all the three Turbulence models (RANS (k-ω SST, K-ε) and LES). 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.

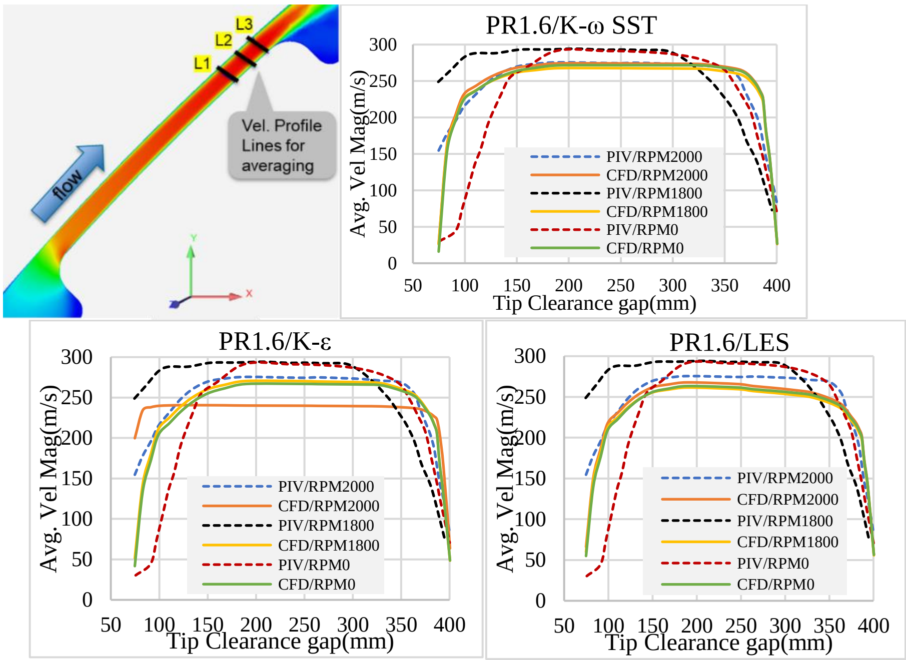


Fig 3. Velocity Profiles comparisons K-ω SST vs. K-ε and LES

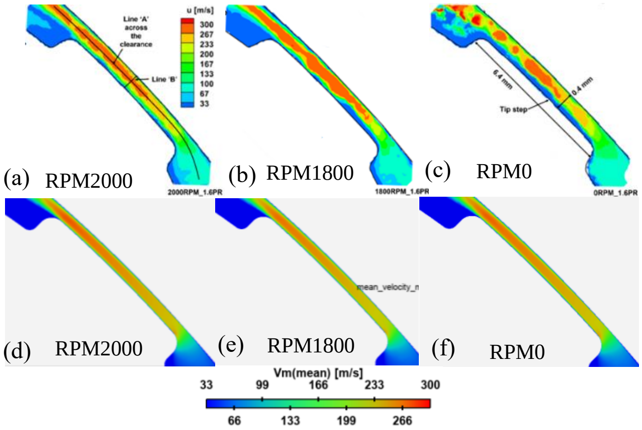


Fig 4. Vel. contours comparison b/w PIV (a)-(c) and CFD k-ω SST model (d)-(f)

For the PR1.4, PIV resulted in the similar trend as PR1.6. 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.

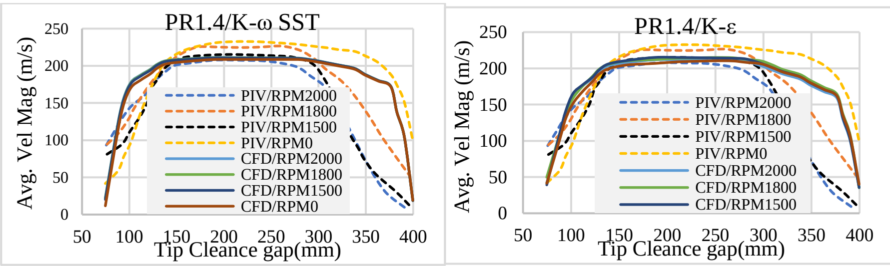


Fig 5. Velocity Profiles comparisons K-ω SST vs. K-ε

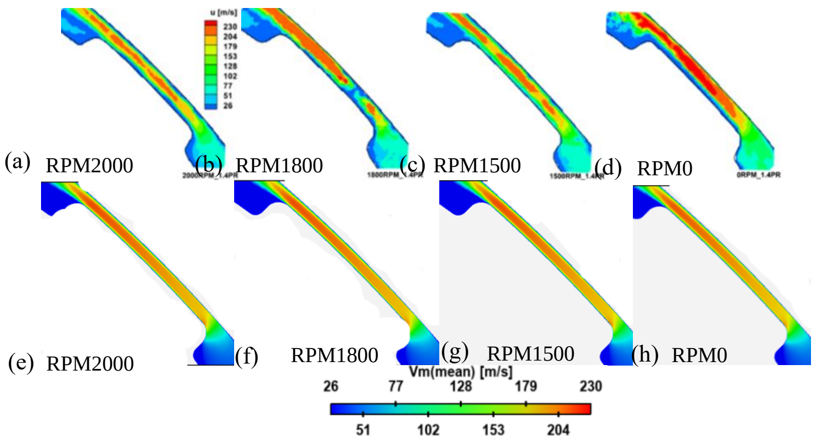


Fig 6. Vel. contours comparison b/w PIV (a)-(d)) and CFD k-omega SST (e)-6(h)

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

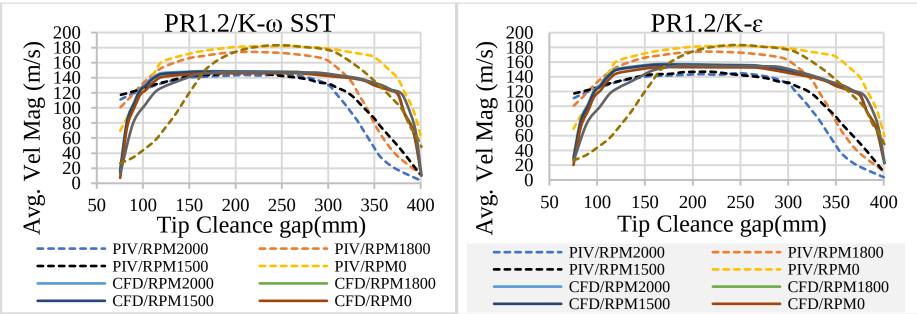


Fig 7. Velocity Profiles comparisons K-ω SST vs. K-ε

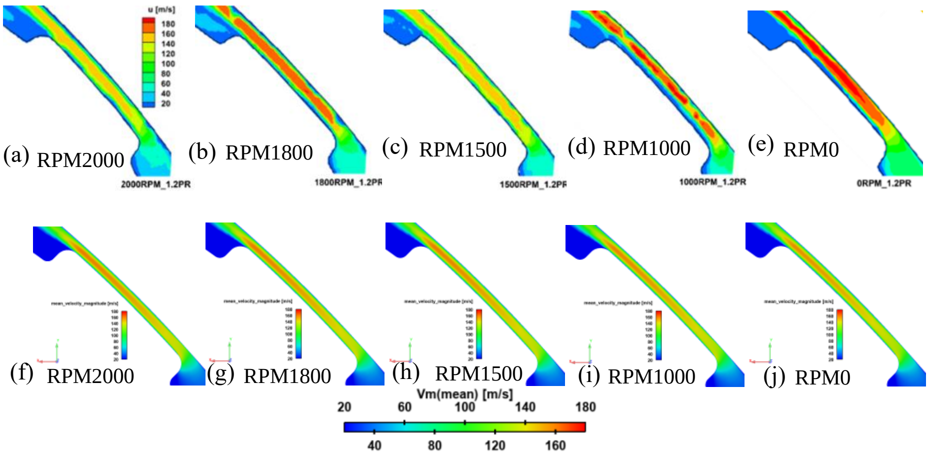


Fig 8. Vel. contours comparison b/w PIV (a)-(e) and CFD k-omega SST (f)-(j)

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

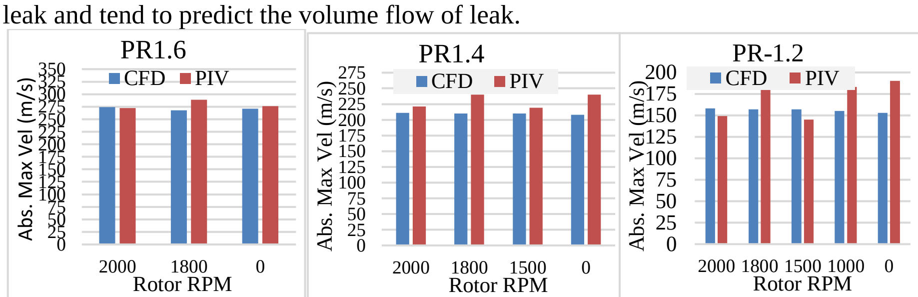


Fig 9. Comparison of absolute max. vel at the tip exit

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

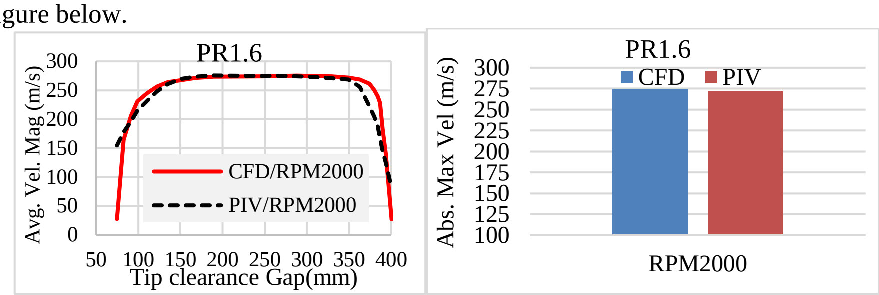
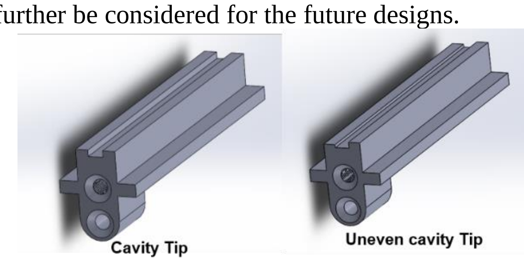


Fig 10. Comparison of Averaged vel. Profile through leakage & absolute max. vel at the tip exit

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 corresponding results will be discussed and more efficient design will further be considered for the future designs.



4.3 Rotor Tip Design Study

Three concepts of Tip are being analyzed using Even Cavity and Uneven Cavity. Fig.11 (b)-type Even cavity tip, Fig11(c)-type Uneven cavity tip with high tip at inlet side and Fig. 11(d)-type Uneven cavity tip with high at exit side.

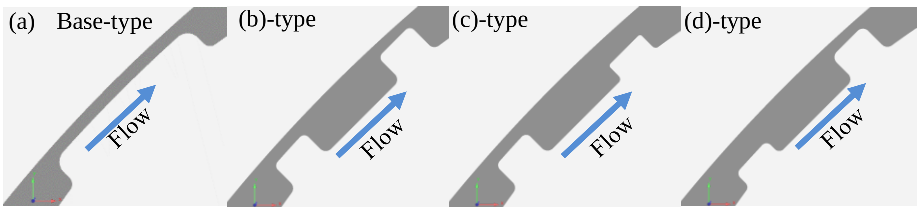


Fig 11. Computational Analysis domains of Tip concepts(2D)

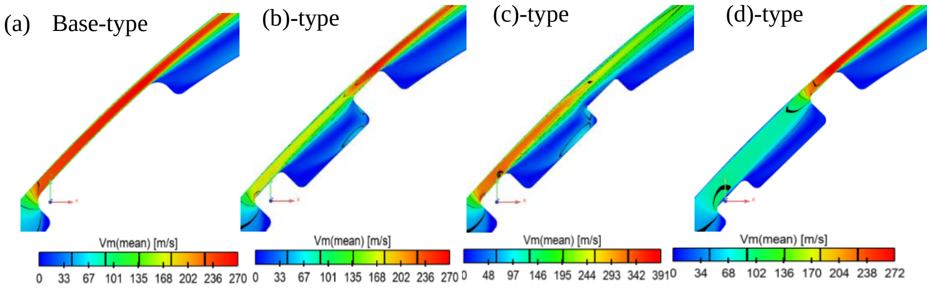


Fig 12. Comparison of mean velocity magnitude contour through the tip gaps

The result shows (Fig. 13(b) below) that Equal cavity tip concept has least leakage (improved by 21%) in the gap compared to other concepts. 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 which has decreased the leakage 15% compared to base. The sudden restriction of flow after the bigger entrance of flow was able to reduce the flow at tip exit. The (c)-type concept has increased the flow speed at entrance and finds wider passage at the exit tip to the downstream. This result will help in designing cavity tip for improved leakage.

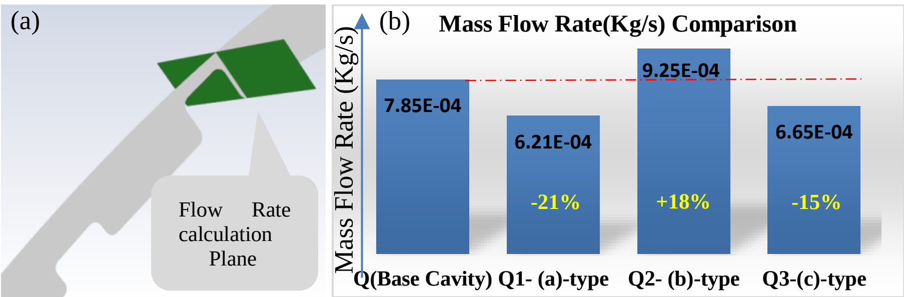


Fig 13. Mass flow rate result comparison between Cavity tip concepts

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 are used for validation with PIV results and analysed. 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 analysed, and suitable geometry to decrease the leakage was found. In future, it is intended to test more cavity tip concepts.

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 [5, 7] 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, towards the project Smart Efficient Compression: Reliability & Energy Targets (SECRET).

References

1. ANSYS.: Fluent theory guide.
2. Mcdougald S, Imrie BW, Cole BN.: An Investigation of the Volumetric Efficiency of a Roots Blower. Purdue e-pubs, (1974).
3. Riley J.. Blower Noise and Solution.: An Introduction to the A.W. Convel Blower. Nov 08 (2017).
4. Sun S, Kovacevic A, Bruecker C, Leto A, Singh G, Ghavami M.: Numerical and Experimental Analysis of Transient Flow in Roots Blower. Feb 06,;4(1):3.(2020).
5. Matuzović M, Rane S, Patel B, Kovačević A, Tuković Ž.: Analysis of conjugate heat transfer in a roots blower and validation with infrared thermography. International Journal of Thermofluids. Nov 16:100234. (2022)
6. Patel B, Kovacevic A, Krupa A. On Measuring Velocity and Temperature in Leakage Flows of Oil Free Rotary Positive Displacement Machines. In: New Technologies, Development and Application IV. Cham: Springer International Publishing; p. 763-73. (2021)
7. Rane S, Kovačević A, Stošić N, Smith IK. Bi-Directional System Coupling for Conjugate Heat Transfer and Variable Leakage Gap CFD Analysis of Twin-Screw Compressors. IOP conference series. Materials Science and Engineering. Sep 01;1180(1):12001(2021).