International Journal of Advanced Technology in Engineering and Science
Vol. No. 10, Issue No. 06, June 2022
www.ijates.com
ijates
ISSN 2348 - 7550

Design and Modelling of a Lobe pump

Joydip Paul, Hiranmoy Samanta*, Soumyodeep Mukherjee, Somnath Mitra, Saikat Acharyya

Gargi Memorial Institute of Technology

Abstract:

Computational Fluid Dynamics (CFD) has been used to design and evaluate amazing displacement machines in recent years. Models based on computational fluid dynamics. In addition, the complex flowing fluid (i.e., the refrigerant) is used in a variety of situations, including near the critical factor and near the saturated-vapor line. In such circumstances, the correct fuel line version no longer holds, necessitating the use of a compressible actual fuel line solution. The custom pre-set mesh technology is one of the most often utilised numerical strategies among the many that have been developed over the years. In this method, a set of hard and fast meshes is constructed prior to running the CFD simulation. The mesh is given to the solver for every step, with the mesh configuration remaining intact. SCORG-V5.2.2 was used to create the meshes of the deforming area around rotating components of the machines in this project. This became linked to OpenFOAM-v1606+, which is utilised to compute the drift discipline associated with the two-lobe Roots blower's operation. It was demonstrated that the proposed technique allows for speedy simulation and excellent agreement with experimental test outcomes.

I. Introduction:

Positive displacement (PD) machines are used in a variety of applications in modern engineering. Numerical techniques are sometimes the simplest way to analyse the machine's capabilities conduct with new fluids without making significant changes to the plant. It should be noted, however, that this type of numerical research is quite difficult. The simulation's complexity has necessitated the use of software that employs a variety of numerical methodologies to clarify the volumetric equipment's operation.When the dynamic motion solver calculates the internal node displacement using the so-called rotor-to-case approach for boundary discretization, cells can collapse or deform into invalid shapes. The most effective way to resolve this is to use the custom predefined mesh era: a set of meshes is ready advanced, reflecting the manage factors by which mesh nodes are skipped. The mesh deformation can be regulated in this way. Grid generation is entirely dependent on decomposing the running area using analytical transfinite interpolation and differential smoothing. This strategy is now widely used in both business and academics. On these artworks, the custom specified mesh era set of rules is utilised.

II. Design of the Roots blower:

Design and assembled in solid works 2016

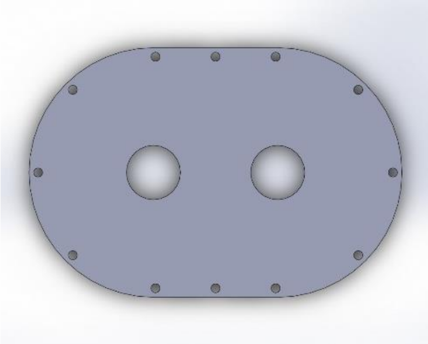

Fig.1-Base

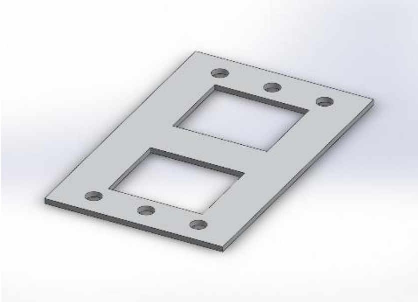

Fig.2-Plate

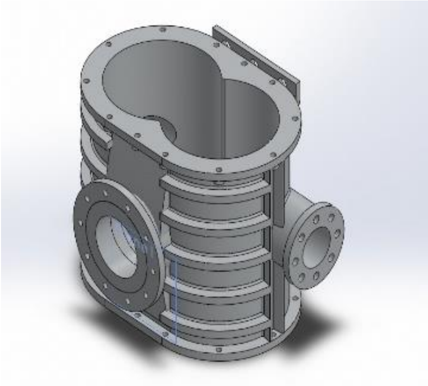

Fig.4-Shell

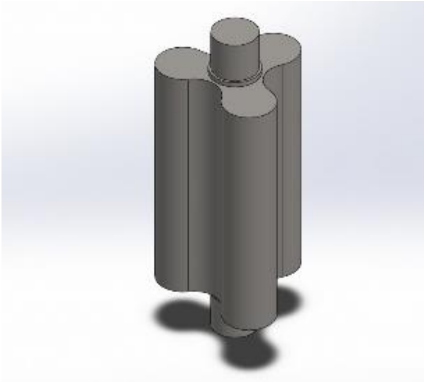

Fig.3-Rotor

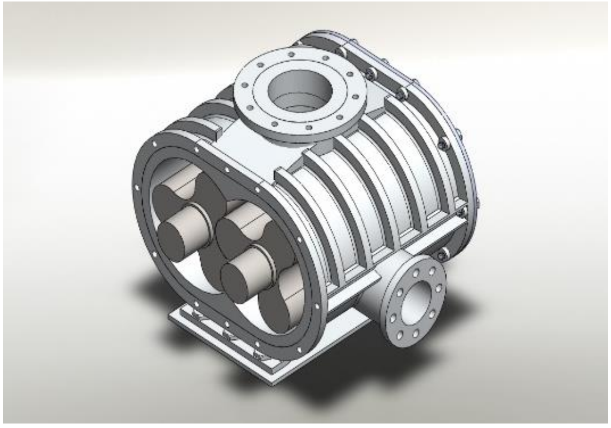

Figure 5: Roots blower (The final product)

III. Method:

The design conditions of the two-lobe roots blower were numerically analysed in this study. The numerical structure and algorithms for moving stitches are discussed in this section.

Mesh:

Rane et al. (2017) have prsented an analysis example of an oil injection twin screw compressor with a new PDE elliptical grid, inter lobe space, radial tip, and rotor core. In the rotor area, a pure hexahedral mesh is mixed with a Cartesian mesh at the inlet and exit ports. The created mesh has the same quality as the original model, so you can run the simulation without making any changes to it. An optional mesh interface (AMI) was used for the interaction between the two lobes. Because the grid is non-conformal, this boundary type is required. As a result, the nodes on both sides of the interface are misaligned. This necessitates additional processing.

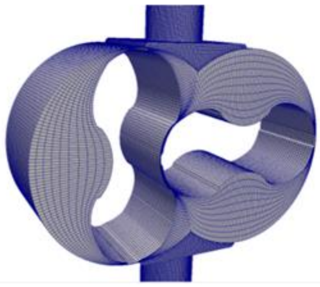

Figure 6: Mesh imported into Open-FOAM-Particular of the lobe area

IV. Numerical set-up:

The boundary conditions employed in the numerical simulation are listed in Table 1. At 1800 rpm, a pressure difference of 0.14 bar was applied between the inlet and outflow. Turbulence is also included, demonstrating that the calculations are sound and can explain the flow's features. High-Re k with standard wall functions are used in the turbulence model. In the whole range of, the fluid to be processed is air, and the compression ratio is near to one. As a result, for this work, we employed the ideal gas model. With four cores, the simulation runs totally in parallel.

Table1: Boundary conditions for the numerical simulation. No Gradient BCs are Neumann-type conditions

Quantity	Inlet	Outlet
Pressure[bar]	1.01	1.15
Temperature[K]	293	No Gradient
Velocity[m/s]	No Gradient	No Gradient
Turbulent kinetic energy	1%	No Gradient
Turbulent kinetic energy dissipation rate	Mixing Length 0.0021 m	No Gradient

V. CFD solution:

The PIMPLE technique is used to integrate velocity and pressure in this solver. The SIMPLE approach (Patankar, 1980) is used in this algorithm, which involves a collection of PISO collectors (Issa, 1986). High Ma currents can make the solver unstable, making it impossible to solve the issue of motion. Instead, utilise the equation of state to link density and pressure. The fluid processed by the equipment in this work is near-standard temperature air. It is plausible to believe that the ideal gas law holds under such conditions. As a result, we'll employ this model for this assignment.

VI. Results:

In this circumstance, the opposing float operates to delay the chamber's release, which is primarily clean (see Figure 7b). There, the lobe on the other side of the chamber with respect to the discharge port has a better strain than the whole region, and the float is flowing from the hole location within side the chamber, as if it were becoming within side the inlet location. The collection of photos presented in Figure 8 demonstrates this.

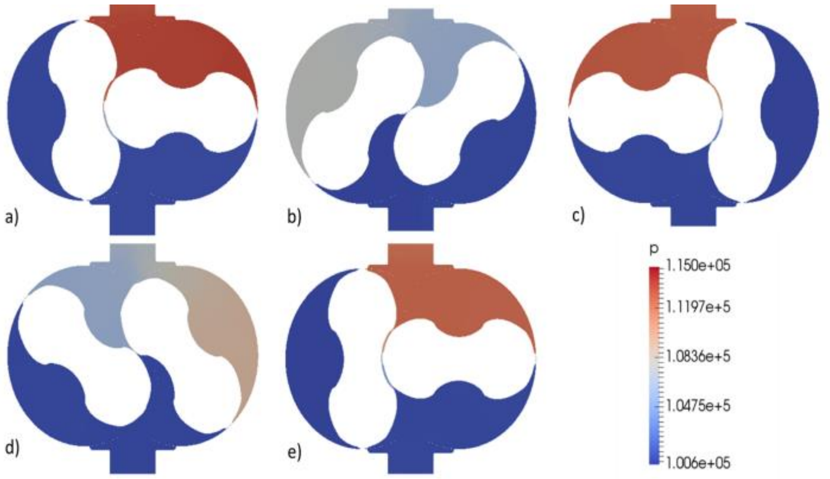

Figure 7: Evolution of the pressure over one half revolution of the rotor

Beginning with a), the lobe tip is approaching the hole port opening. Because of the current discharge section of the left chamber and the glide returning from the hole, the fluid systems on the left-hand side of the rotor are quite uneven. b) At the outlet, the over-stress discharge region tries to force the circulation into the proper chamber. This "backflow-jet" is sensed for a brief angular period, and a new vortex forms in c) as the rotor attempts to discharge the glide trapped within the correct chamber in the direction of the hole port/region.

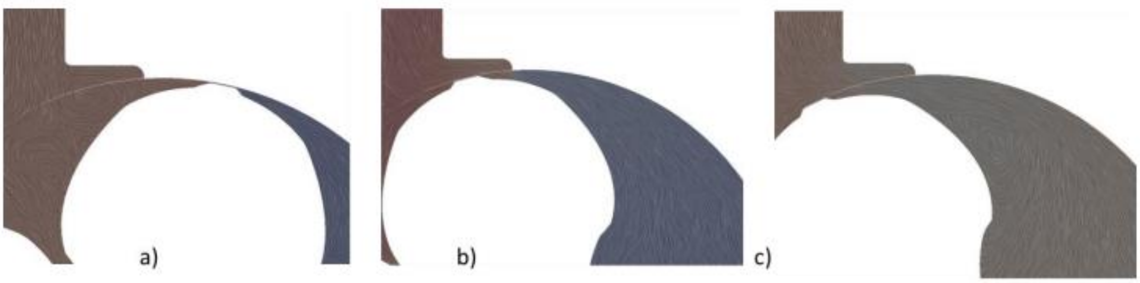

Figure 8: Vortex structures at the chamber opening

It will be interesting to witness how the two rotors react to each other's blowback. They can clearly see that the area's normal structure is shown accurately and that there are no interface-related discontinuities. The task's solution simply addresses the extremely high speeds reached in the gap. When investigating positive displacement machines, simulation helps you to observe features of general interest. The distance between the rotor and the housing is one of the most important characteristics for machine performance, as detailed in

(Casari, 2017) and (Casari, 2018). (Suman, 2017). The volumetric efficiency of the machine is driven by such gaps, as well as the spacing between rotors.

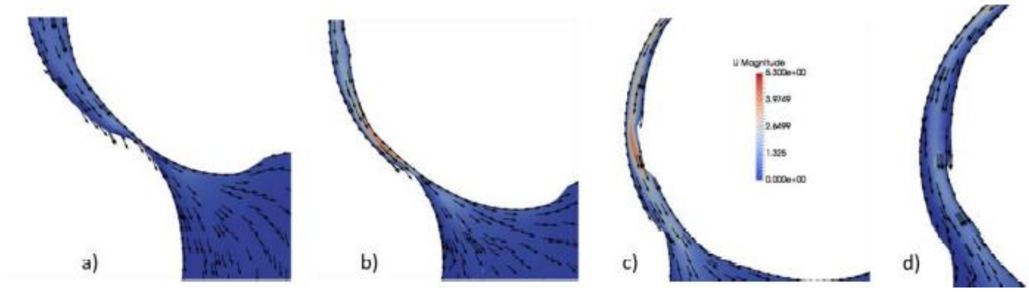

Figure 9: Flow field evolution in the clearance between the rotors

VII. Conclusion

The position of the node is automatically selected in time steps using linear interpolation between two successive control points. This approach has been demonstrated to be reliable, and it can be used to simulate particularly aggressive counter-current circumstances.

As a result, the most important results of this work are:

• Development of a dynamic mesh library capable of studying the factors generated from SCORG for riding the mesh corresponding mesh motion;

• Simulation of a real advantageous displacement machine, with hole lengths that can be close to the not uncommon place engineering practise;

• The dynamic mesh method used without violating the distance conservation law, implying the non-conservativeness of the mass.

References:

1. Blekhman, D. I., Mollendorf, J. C., Felske, J. D., Lordi, J. A., & Joshi, A. M. (2004). Roots Compressor: High Temperature Testing and Modeling. In ASME 2004 International Mechanical Engineering Congress and Exposition (pp. 43–58). American Society of Mechanical Engineers.
2. Casari, N. Suman, A., Morini, M. & Pinelli, M. (2017). Real Gas Expansion with Dynamic Mesh in Common Positive Displacement Machines. Energy Procedia, 248--255.
3. Casari, N., Suman, A., Ziviani, D., van den Broek, M., De Paepe, M., & Pinelli, M. (2017). Computational Models for the Analysis of positive displacement machines: Real Gas and Dynamic Mesh. Energy Procedia, 129, 411–418.
4. Chang, J.-C., Chang, C.-W., Hung, T.-C., Lin, J.-R., & Huang, K.-C. (2014). Experimental study and CFD approach for scroll type expander used in low-temperature organic Rankine cycle. Applied Thermal Engineering, 73(2), 1444–1452.

5. Gonzalez, E. (2016). Cfd simulations of acoustic and thermoacoustic phenomena in internal flows. 46th AIAA Fluid Dynamics Conference, (p. 3960).

6. Issa, R. I. (1986). Solution of the implicitly discretised fluid flow equations by operator-splitting. Journal of computational physics, 40-65.

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

8. Kang, Y.-H. a.-H.-H. (2012). Factors impacting on performance of lobe pumps: A numerical evaluation. Journal of Mechanics, 229--238.

9. Kovacevic, A., Stosic, N., & Smith, I. (2007). Screw compressors: three dimensional computational fluid dynamics and solid fluid interaction (Vol. 46). Springer Science & Business Media.

10. Kovačević, A, Rane S. (2017). Algebraic generation of single domain computational grid for twin screw machines Part II – Validation, Advances in Engineering Software, 107, pp., doi: 10.1016/j.advengsoft.2017.03.001

11. Mittal, R., & Iaccarino, G. (2005). Immersed boundary methods. Annu. Rev. Fluid Mech., 37, 239–261.

12. Morini, M., Pavan, C., Pinelli, M., Romito, E., & Suman, A. (2015). Analysis of a scroll machine for micro ORC applications by means of a RE/CFD methodology. Applied Thermal Engineering, 80, 132–140.

13. Patankar, S. (1980). Numerical heat transfer and fluid flow. CRC press.

14. Rane, S., Kovacevic, A., Stosic, N., & Kethidi, M. (2013). Grid deformation strategies for CFD analysis of screw compressors. International Journal of Refrigeration, 36(7), 1883–1893.

15. Rane, S., Kovačević, A. (2017). Algebraic generation of single domain computational grid for twin screw machines. Part I. Implementation, Advances in Engineering Software, 107, pp. 38-50.

16. Suman, A., Randi, S., Casari, N., Pinelli, M., & Nespoli, L. (2017). Experimental and Numerical Characterization of an Oil-Free Scroll Expander. Energy Procedia, 129, 403–410.

17. Compact design Roots Blower Manufacturers, Suppliers from Nantong RONGHENG (rhonorblower.com)

18. D.C.H. Yang, S.H. Tong, The specific flowrate of deviation function based lobe pumps—derivation and analysis, Mech. Mach. Theory 37 (2002) 1025–1042.

19. S.H. Tong, D.C.H. Yang, Rotor profiles synthesis for lobe pumps with given flow rate functions, ASME J. Mech. Des. 127 (2) (2005) 287–294.

20. Y.W. Hwang, C.F. Hsieh, Study on high volumetric efficiency of the Roots rotor profile with variable trochoid ratio, Proc. Inst. Mech. Eng. C J. Mech. Eng. Sci. 220 (9) (2006) 1375–1384.

21. C.F. Hsieh, Y.W. Hwang, Study on the high-sealing of Roots rotor with variable trochoid ratio, ASME J. Mech. Des. 129 (2007) 1278–1284. [15]

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

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

24. J.V. Voorde, J. Vierendeels, E. Dick, Flow simulations in rotary volumetric pumps and compressors with the fictitious domain method, Comput. Methods Appl. Mech. Eng. 168 (2004) 491–499.

25. G. Houzeaux, R. Codina, A finite element method for the solution of rotary pumps, Comput. Fluids 36 (4) (2007) 667–679.

26. W. Strasser, CFD investigation of gear pump mixing using deforming/agglomerating mesh, ASME J. Fluids Eng. 129 (4) (2007) 476–484.

27. Z.F. Huang, Z.X. Liu, Numerical study of a positive displacement blower, Proc. Inst. Mech. Eng. C J. Mech. Eng. Sci. 223 (10) (2009) 2309–2316.