2016

CFD Simulation of An Oil Flooded Scroll Compressor Using VOF Approach

Hui Ding
Simerics Inc., United States of America, hd@simerics.com

Yu Jiang
Simerics Inc., United States of America, yj@simerics.com

CFD Simulation of An Oil Flooded Scroll Compressor Using VOF Approach

Hui Ding*, Yu Jiang

Simerics Inc.,
1750 112th Ave NE Ste A203,
Bellevue, WA 98004, USA
Phone – 425-502-9978, Fax – 425-502-9978, E mail – hd@simerics.com

* Corresponding Author

ABSTRACT

Liquids have been commonly used in different types of compressors to cool the compressed gas and to seal the leakage gaps in order to increase the efficiency. CFD simulation provides valuable insights to help design engineers to verify, to analyze, and to improve the performance of a compressor. However two phase flow with moving parts and small gaps is a very challenging CFD problem. For compressor simulation, thermal effects and heat transfer are also essential. Therefore simulation of liquid flooded compressors is extremely difficult.

In this paper a full 3D transient CFD model for a generic oil flooded scroll compressor will be described in detail. Volume Of Fluid (VOF) multiphase approach will be used to model gas and liquid phases. Effects of flooded oil will be evaluated by comparing the simulation results for the cases with and without oil. Simulation will also demonstrate that the approaches used in the paper are robust, fast, and user friendly, and can be readily applied to industrial compressor systems.

Keywords: oil flooded, scroll compressor, CFD, two phase flow, VOF.

1. INTRODUCTION

Compressors are designed to compress gases. However, liquids, such as oil and water, are often introduced to compression process for various purposes. For example, oil in compressor not only lubricates the moving components, but also cools the compressed gas, and seals the leakage gaps to help improve the efficiency of the process (Bell, 2011).

In recent years, CFD has been widely used to analyze performance of varies Positive Displacement (PD) compressors. Those models cover different types of PD compressors including rolling piston compressor (Lenz and Cooksey, 1994, Geng *et al.*, 2004, Liang, *et al.*, 2010, Ding and Gao, 2014,), scroll compressor (Feng et al., 2004, Cui, 2006, Gao and Jiang 2014, Gao, *et al.*, 2015), twin screw compressor (Voorde *et al.*, 2005, Kovacevic *et al.*, 2007, Pascu *et al.*, 2012, Kovacevic *et al.*, 2014), and reciprocating compressor (Birari *et al.*, 2006, Pereira *et al.*, 2010, Dhar *et al.*, 2016). Intake and/or discharge valves are also included in some of the models. However almost all of the models are running as a "dry" process, meaning no liquid modeled in the compression chamber. The lack of CFD model of liquid flooded PD compressors is mainly due to the difficulties in CFD multiphase flow simulation.

Even for a relatively simple two phase problem, high density ratio between liquid and gas, sophisticated interaction among the phases, and the interface tracking with complex shape make the flow difficult to solve. It becomes even more difficult to solve a multiphase flow in a PD compressor. In such a case, gas phase has to be treated as compressible, heat transfer is also essential, and interface tracking has to be done in moving, deforming volumes. The major issues users experience with many CFD solvers in multiphase simulations are poor convergence, very long simulation time, and unsatisfactory mass/energy conservation.

Simerics-PD (also known as PumpLinx) has been designed to model PD machines including PD compressors. Its VOF based multiphase model has also been validated with many different industrial applications including, liquid ring vacuum pump (Ding et al., 2015), and oil pump priming (Kucinschi and Shieh, 2016). Recently, heat transfer capability was added to Simerics-PD's VOF multiphase model. Oil flooded PD compressor is one of the targeted application for this new capability.

Scroll compressors are widely used in many industries, such as refrigeration, air-conditioning and automotive. It is believed that scroll compressors have the advantages of high efficiency, lower noise and vibration levels. Scroll compressor can run dry or oil flooded, and it was chosen as the candidate for this study.

This paper presents a full 3D transient CFD model for a generic oil flooded scroll compressor. Due to the time limitation, the focus of this paper will be on the demonstration of the new capability with emphasis on the qualitative trends revealed from simulation results, and the conservation of mass and energy in the results.

2. CFD SOLVER AND GOVERNING EQUATIONS

2.1 Conservation Equations For Gas Liquid Mixture

The CFD package used in this study solves conservation equations of mass, momentum, and energy of a compressible fluid using a finite volume approach. Those conservation laws can be written in integral representation as

$$\frac{\partial}{\partial t}\int_{\Omega(t)}\rho d\Omega+\int_{\sigma}\rho(\boldsymbol{v}-\boldsymbol{v}_{\sigma})\cdot\boldsymbol{n}d\sigma=0 \tag{1}$$

$$\frac{\partial}{\partial t}\int_{\Omega(t)}\rho\boldsymbol{v}d\Omega+\int_{\sigma}\rho\big((\boldsymbol{v}-\boldsymbol{v}_{\sigma})\cdot\boldsymbol{n}\big)\boldsymbol{v}d\sigma= \int_{\sigma}\tilde{\boldsymbol{\tau}}\cdot\boldsymbol{n}d\sigma-\int_{\sigma}p\boldsymbol{n}d\sigma+\int_{\Omega}\boldsymbol{f}d\Omega \tag{2}$$

$$\frac{\partial}{\partial t}\int_{\Omega(t)}\rho Ed\Omega+\int_{\sigma}\rho\big((\boldsymbol{v}-\boldsymbol{v}_{\sigma})\cdot\boldsymbol{n}\big)Ed\sigma= \int_{\sigma}k\nabla T\cdot\boldsymbol{n}d\sigma-\int_{\sigma}p\boldsymbol{v}\cdot\boldsymbol{n}d\sigma+\int_{\sigma}(\boldsymbol{v}\cdot\tilde{\boldsymbol{\tau}})\cdot\boldsymbol{n}d\sigma+\int_{\Omega}\boldsymbol{f}\cdot\boldsymbol{v}d\Omega \tag{3}$$

The standard $k-\varepsilon$ two-equation model (Launder and Spalding, 1974) is used to account for turbulence,

$$\frac{\partial}{\partial t}\int_{\Omega(t)}\rho kd\Omega+\int_{\sigma}\rho\big((\boldsymbol{v}-\boldsymbol{v}_{\sigma})\cdot\boldsymbol{n}\big)kd\sigma= \int_{\sigma}(\mu+\frac{\mu_t}{\sigma_k})(\nabla k\cdot\boldsymbol{n})d\sigma+\int_{\Omega}(G_t-\rho\epsilon)d\Omega \tag{4}$$

$$\frac{\partial}{\partial t}\int_{\Omega(t)}\rho\varepsilon d\Omega+\int_{\sigma}\rho\big((\boldsymbol{v}-\boldsymbol{v}_{\sigma})\cdot\boldsymbol{n}\big)\varepsilon\, d\sigma= \int_{\sigma}(\mu+\frac{\mu_t}{\sigma_\varepsilon})(\nabla\epsilon\cdot\boldsymbol{n})d\sigma+\int_{\Omega}(c_1G_t\frac{\varepsilon}{k}-c_2\rho\frac{\varepsilon^2}{k})d\Omega \tag{5}$$

Together with equation of state, where properties are functions of temperature and pressure, to form a closed system:

$$\rho=f(p,T) \tag{6}$$

In the solver, each of the fluid properties can be a function of local pressure and temperature, and can be input as a formula or in a table format.

2.2 VOF Model for Multiphase

VOF models are widely used in simulation of two phase flow (Ubbink 1977, Hirt and Nichols, 1981) . VOF solves a set of scalar transport equations representing the fraction of the volume each fluid component occupies in every computational cell. In the region close to a sharp interface, reconstruction will be used to determine the shape of the interface. The transport equation of the volume fraction for each fluid component can be written as:

$$\frac{\partial}{\partial t}\int_{\Omega(t)} \rho_i F_i d\Omega + \int_{\sigma} \rho_i (\boldsymbol{v} - \boldsymbol{v}_\sigma) \cdot \boldsymbol{n} F_i d\sigma = 0 \tag{7}$$

Where F_i is the volume fraction of the *ith* fluid component, and ρ_i is the local density of *ith* fluid component. The weighted mixture density of the fluid in equation (1) to (5) are then calculated as:

$$\rho = \sum \rho_i F_i \tag{8}$$

Both implicit and explicit methods are implemented to solve this equation. Close to the sharp phase interface, high resolution scheme was implemented for reconstruction of the interface.

This software package has been validated against many different types of compressors including: centrifugal compressor, lobe compressor, twin screw compressor (Kovacevic1 *et al*., 2014), scroll compressor (Gao and Jiang, 2014, Gao et al., 2015), rolling piston (Ding and Gao, 2014), and reciprocating compressor (Dhar et al., 2016) for single phase compression of air, refrigerants, and other type of gases. The VOF model has also been validated against many industrial applications (Ding et al., 2015, Kucinschi and Shieh, 2016) for multiphase flow without solving heat transfer directly.

3. SCROLL COMPRESSOR TEST CASE

A generic scroll model was used to demonstrate the functionality and capability of proposed approach. The complete system includes an inlet port, a scroll, and an outlet port. Mesh of the scroll was created using Simerics-PD Scroll Template (Gao and Jiang, 2014). The rest of fluid volumes are meshed using Simerics binary tree unstructured mesher. All the fluid volumes are connected together using Miss Matched Grid Interface (MGI). The total number of cells is around 0.3 million. Figure 1 shows the complete fluid domain. Figure 2 shows the mesh in a cutting plane.

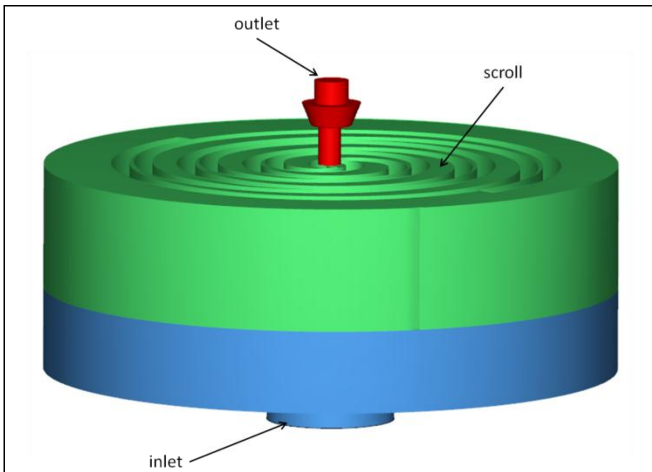


Figure 1: All fluid volumes

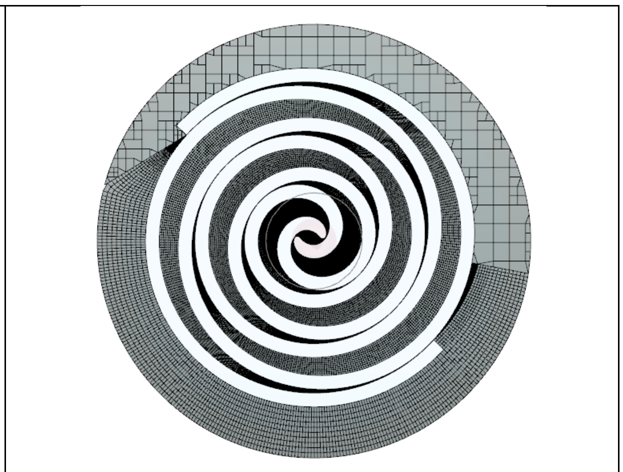


Figure 2: Mesh in a cutting plane

The inlet is set to a fixed pressure, fixed temperature boundary condition. The outlet is set to a fixed pressure boundary condition. The refrigerant is R410a, modeled using ideal gas law. The molecular weight of R410a is 72.63 g/mole, and the heat capacity is 1035 J/kgK. The compressor rotation speed is 3500 RPM. The oil was assumed to be incompressible with a density of 800 Kg/m3, and a heat capacity of 1670 J/kgK. In order to demonstrate the effects of the flooded oil, a similar case with the same parameters but without oil was also simulated for comparison.

Start from properly prepared CAD geometry, the meshing and the setup of the simulation take less than half an hour with the help of the Scroll template. Simulation time is about 1.5 hours per revolution for oil flooded simulation and about 20 minutes per revolution without oil on a PC with quad-core Intel Xeon CPU at 2.67GHz.

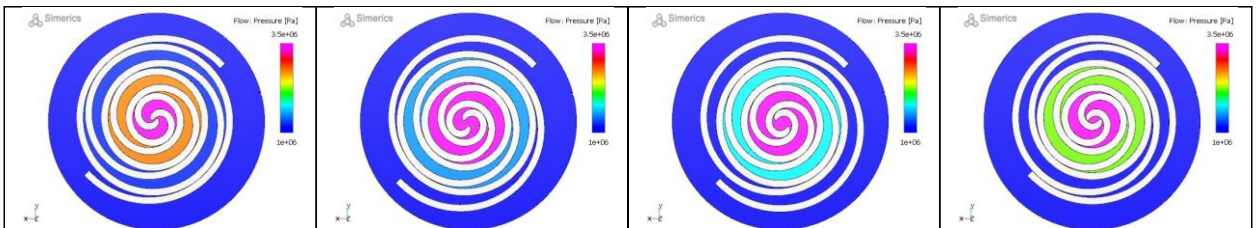
4. RESULTS AND DISCUSSION

In the simulation, the inlet and the outlet pressure are set to 1MPa and 3.4MPa respectively. The inlet temperature is set to 300K. The oil is assumed uniformly mixed with refrigerant at the entrance. Two oil concentrations were simulated. The oil mass fraction were set to 2.4% and 18.1% respectively. The corresponding inlet oil volume fraction were about 0.086% and 0.65%. Simulation results start to stabilize after around 6 revolutions. Table 1 shows the mass and energy imbalance for the three simulations.

Table 1 Conservation of simulation results

	Oil flooded (2.4%)	Oil flooded (18.1%)	Dry
Gas mass imbalance	1.2%	2.0%	0.2%
Oil mass imbalance	0.7%	1.2%	N/A
Energy imbalance	1.3%	1.8%	0.5%

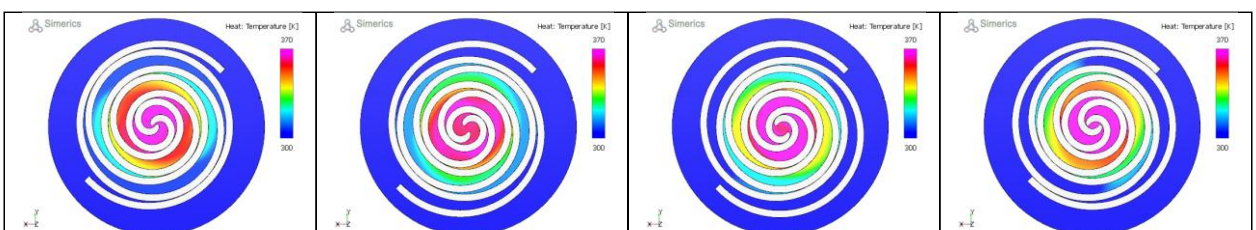
Figure 3 shows typical pressure contour at 4 different crankshaft angles. The pressure in each isolated fluid "pocket" keeps increasing with crankshaft angle due to the continuous volume reduction of the pocket till it reaches outlet.



90 degree	180 degree	270 degree	360 degree

Figure 3: Pressure contour at different crankshaft angles

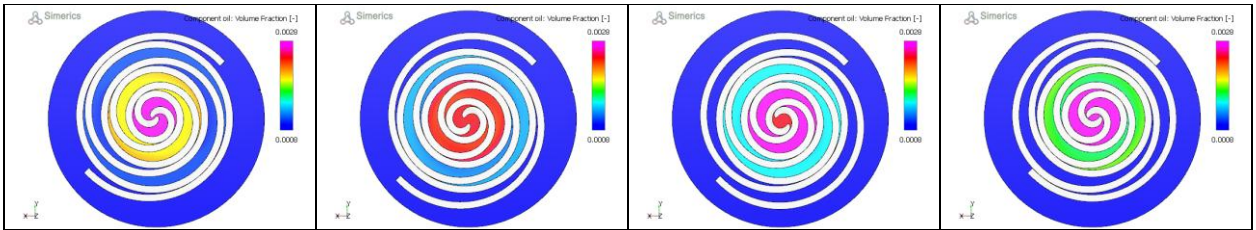
Figure 4 shows typical temperature contour at 4 crankshaft angles. Temperature in the pocket follows a similar trend. However unlike the pressure, the temperature inside each pocket is not very uniform. The non-uniformity is caused by the leakage flow from the high pressure/high temperature region towards the low pressure/low temperature region. Pressure propagates with pressure wave in the speed of sound, while temperature propagates with much slower convection and diffusion process. It is expected that the pressure in each chamber equalizes much fast than the temperature.



90 degree	180 degree	270 degree	360 degree

Figure 4: Temperature contour at different crankshaft angles

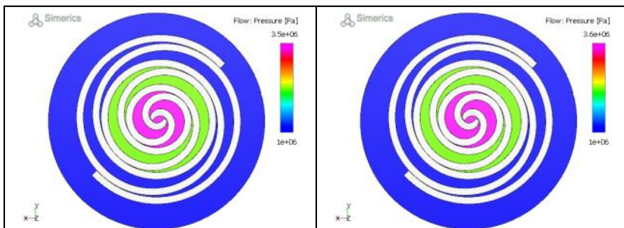
Figure 5 shows oil volume fraction contour for 2.4% oil case at 4 crankshaft angles. When pocket pressure increases, volume of the gas phase will decrease, and the volume fraction of oil will increase. Therefore oil volume fraction also increase when pocket moves towards the center.



90 degree	180 degree	270 degree	360 degree

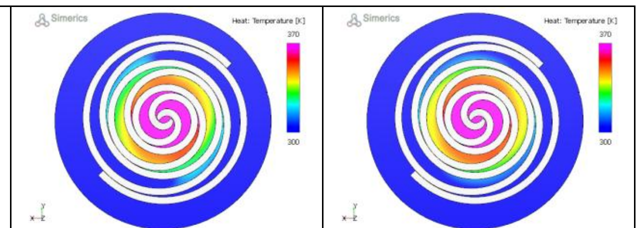
Figure 5: Oil volume fraction at different crankshaft angles

Figure 6 and Figure 7 compare the pressure and the temperature for the cases with and without oil at the same crankshaft angle. Although the pressure distribution looks very similar, the temperature for the dry compression is apparently higher.



Oil flooded	Dry

Figure 6: Pressure with/without oil



Oil flooded	Dry

Figure 7: Temperature with/without oil

Figure 8 and Figure 9 plot the pressure and the temperature history of a single fluid pocket for three simulated cases. Those data are read from monitor points moving together with the pocket. From the plots, the pressure history are similar for all the cases, and the 2.4% oil flooded case is almost identical to the dry case. But there are much more differences in temperature history. The temperature for the case with 18.1% oil dropped about 20 degree at the outlet.

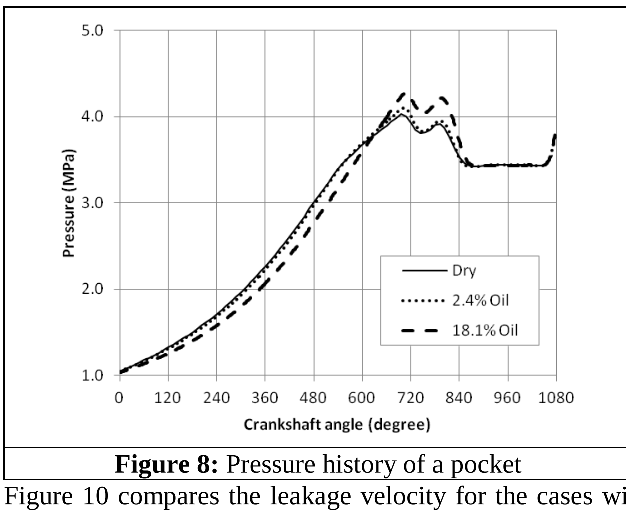


Figure 8: Pressure history of a pocket

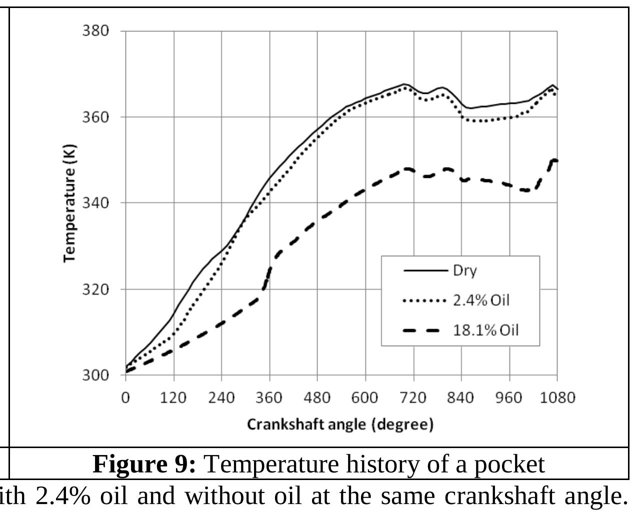
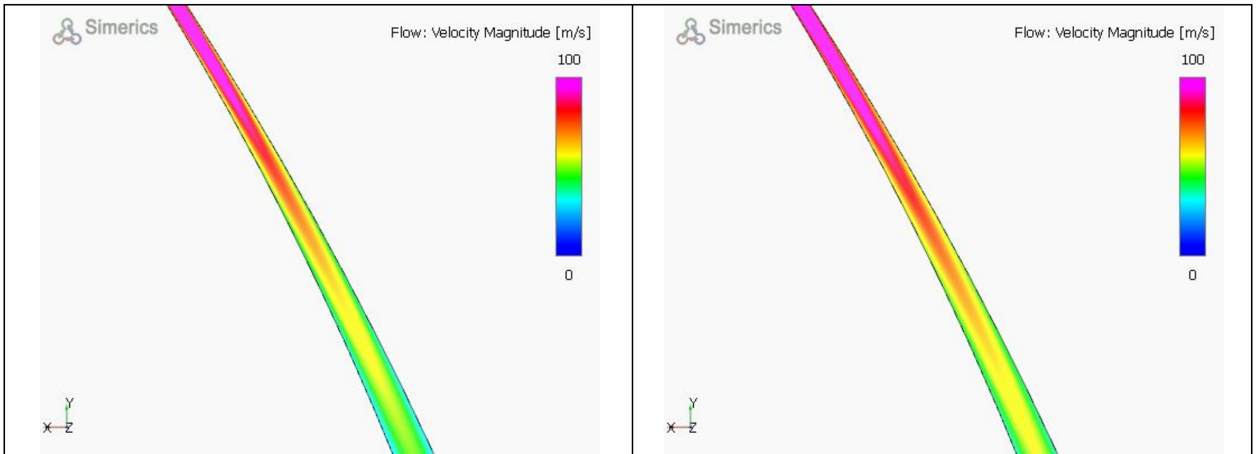


Figure 9: Temperature history of a pocket

Figure 10 compares the leakage velocity for the cases with 2.4% oil and without oil at the same crankshaft angle. The leakage velocity for the dry compression is higher than the oil flooded one.



Oil flooded	**Dry**

Figure 10. Leakage flow velocity with/without oil

Table 2 shows the major performance difference when compressor run dry or with oil flooded. The results show a gradual increase in refrigerant mass flow rate, a gradual decrease in power consumption, and a significant temperature reduction when the oil contents in compressor increases. Those trends match the expected oil cooling and sealing effects.

Table 2 Differences in compressor performance

Oil flooded/Dry	2.4% Oil	18.1% Oil
Gas mass flow rate	101.3%	107.1%
Outlet temperature rise	90.4%	63.9%
Rotor power	99.5%	97.6%

5. CONCLUSIONS AND FUTURE WORK

A newly improved VOF multiphase model has been successfully applied to an oil flooded scroll compressor. Simulation results predict correctly the oil cooling and sealing effects on compression process, and show good mass and energy conservation. With the help of template design and robustness of flow solver, the setup and simulation are also easy and fast. The new model has demonstrated great potential for modeling two phase flow problems in PD compressors. Next step will be validation against available test data. Further improvement of simulation speed, and inclusion of phase change model are in the future development plan.

NOMENCLATURE

C_1	Turbulence model constant		T	Temperature	K
C_2	Turbulence model constant		t	Time	s
C_μ	Turbulence model constant		$\boldsymbol{v}$	Velocity vector	
$\boldsymbol{E}$	Total energy	J	*VOF*	Volume of fluid	
$\boldsymbol{F_i}$	Volume fraction of ith component		ε	Turbulence dissipation	m^2/s^3
$\boldsymbol{f}$	Body force	N	μ	Fluid viscosity	Pa-s
G_t	Turbulent generation term		μ_t	Turbulent viscosity	Pa-s
k	Heat conductivity	W/m-K	ρ	Fluid density	kg/m^3
k	Turbulence kinetic energy	m^2/s^2	σ	Surface of control volume	
MGI	Mis-matched grid interface		σ_k	Turbulence model constant	
$\boldsymbol{n}$	Surface normal		σ_ε	Turbulence model constant	
PD	Positive displacement		$\boldsymbol{\tau}$	Stress tensor	

p	Pressure	Pa	Ω	Control volume
RPM	Revolution per minute			

REFERENCES

Bell, I., 2011. Theoretical and Experimental Analysis of Liquid Flooded Compression in Scroll Compressors. Ph.D. thesis, Purdue University. Full-text: http://docs.lib.purdue.edu/herrick/2/

Birari, Y. V., Gosavi, S. S., & Jorwekar, P. P. (2006). Use of CFD in design and development of R404a reciprocating compressor. *Proceedings of the International Compressor Engineering Conference at Purdue.* West Lafayette, IN, USA.

Cui, M.M., 2006, "Numerical Study of Unsteady Flows in a Scroll Compressor", 2006, *J. Fluids Eng.* Vol.128 no.5, p. 947-955.

Dhar, S., Ding, H., and Lacerda, J., 2016, "A 3-D Transient CFD Model of a Piston Reciprocating Compressor with Dynamic Port Flip Valves," *23rd Int. Compressor Engrg. Conf. at Purdue*. Paper 1392.

Ding, H. and Gao, H., 2014,"3-D Transient CFD Model For A Rolling Piston Compressor With A Dynamic Reed Valve", 2014, *22nd International Compressor Engineering Conference*, Paper 1548.

Ding, H., Jiang, Y., Wu, H., and Wang, J., 2015, "Two Phase Flow Simulation of Water Ring Vacuum Pump Using VOF Model," ASME/JSME/KSME 2015 Joint Fluids Engineering Conference, Seoul, South Korea, July 26–31, 2015

Feng, J., Qu, Z., and Lin, X., "Three-Dimension Numerical Simulation of Discharge Flow in a Scroll Air Compressor", 2004. *International Compressor Engineering Conference*. Paper 1647.

Gao, H., and Jiang, Y., 2014, Numerical Simulation of Unsteady Flow in a Scroll Compressor, *Int. Compressor Engrg. Conf. at Purdue*.

Gao H., Ding, H., and Jiang, Y., 2015 "3D Transient CFD Simulation of Scroll Compressors with the Tip Seal,"IOP Conference Series: Materials Science and Engineering. Vol. 90. No. 1. IOP Publishing, 2015.

Geng, W., Liu, C., and Wang, Y., 2004, The Performance Optimization of Rolling Piston Compressors Based on CFD Simulation, *Int. Compressor Engrg. Conf. at Purdue*, pp. 1488-1492.

Hirt, C.W. and Nichols, B.D., 1981, Volume of fluid (VOF) method for the dynamics of free boundaries. J. Comput. Phys., Vol. 39, p. 201-225.

Kinjo, K., Nakano, A., Hikichi, T., and Morinishi, K., 2010, Study of CFD Considering Valve Behavior in Reciprocating Compressor, *Int. Compressor Engrg. Conf. at Purdue*. Paper 1975

Kovacevic A., Stosic N. and Smith I. K., 2007, Screw compressors - Three dimensional computational fluid dynamics and solid fluid interaction, ISBN 3-540-36302-5, Springer-Verlag Berlin Heidelberg New York.

Kovacevic, A., Rane, S., Stosic, N., Jiang, Y., and Lowry, S., 2014, Influence of approaches in CFD Solvers on Performance Prediction in Screw Compressors, *Int. Compressor Engrg. Conf. at Purdue*.

Kucinschi, B. and Shieh, T., "Estimation of Oil Supply Time during Engine Start-Up at Very Low Temperatures," SAE Int. J. Fuels Lubr. 9(2):2016, doi:10.4271/2016-01-0893.

Launder, B.E., and Spalding, D.B., 1974, The numerical computation of turbulent flows, Comput. Methods Appl. Mech. Eng., 3, pp. 269-289.

Lenz, J. R., Cooksey, E. A., 1994, Application of Computational Fluid Dynamics to Compressor Efficiency Improvement, *Int. Compressor Engrg. Conf. at Purdue*, pp. 441-446.

Liang, S., Xia, S., Kang, X., Zhou, P., Liu, Q., and Hu, Y., 2010, Investigation of Refrigerant Flow Simulation and Experiment of Rolling Piston, *Int. Compressor Engrg. Conf. at Purdue*, pp.

Machu, G.,; Albrecht, M., Bielmeier, O., Daxner, T., and Steinruck, P., 2004, A Universal Simulation Tool for Reed Valve Dynamics, *Int. Compressor Engrg. Conf. at Purdue*. Paper 1716

Pascu M., Kovacevic A., and Udo N., 2012, Performance optimization of Screw Compressors based on numerical investigation of the flow behaviour in the discharge chamber. Int Compressor Conf at Purdue, Purdue, p. 1145.

Pereira, E. L., Deschamps, C. J., & Ribas, F. A. (2010). Numerical analysis of heat transfer inside the cylinder of reciprocating compressors in the presence of suction and discharge processes. *Proceedings of the International Compressor Engineering Conference at Purdue.* West Lafayette, IN, USA.

Prasad, B. G. S., 2004, CFD for Positive Displacement Compressors, *Int. Compressor Engrg. Conf. at Purdue*. Paper 1689

Silva, E., Deschamps, C. J., and Fancello , E. A., 2012, A Procedure to Optimize Reed T ype V alves Considering Efficiency and Bending Fatigue, *Int. Compressor Engrg. Conf. at Purdue*. Paper 2106.

Ubbink, O., 1977. “Numerical prediction of two fluid systems with sharp interfaces”. Ph.D. Thesis, Department of Mechanical Engineering, Imperial College, University of London.

Voorde John Vande and Vierendeels Jan, 2005, A grid manipulation algorithm for ALE calculations in screw compressors. 17th AIAA Computational Fluid Dynamics Conference, Canada, AIAA 2005-4701.