2014

Numerical Simulation of Unsteady Flow in a Scroll Compressor

Haiyang Gao
Simerics, Inc, United States of America, hg@simerics.com

Numerical Simulation of Unsteady Flow in a Scroll Compressor

Haiyang GAO[1]*, Yu JIANG[2]

[1]Simerics, Inc.,
1750 112th AVE NE Ste A203
Bellevue, WA 98004, USA
hg@simerics.com

[2]Simerics, Inc.,
1750 112th AVE NE Ste A203
Bellevue, WA 98004, USA

* Corresponding Author

ABSTRACT

A new template simulation tool is developed for scroll compressors/expanders. This scroll template generates a high quality multi-block structured mesh from the outlines of stationary and orbiting rotors, which are inputs from a user. The mesh movement is also automatically calculated to account for every position of the orbiting rotor, maintaining good grid quality and smooth movement throughout the whole revolution. The new approach is demonstrated on a generic scroll compressor with a dynamically controlled reed valve at the discharge port. It is shown that the new template greatly simplifies the setup of the Computational Fluid Dynamics (CFD) simulation and can significantly reduce turn-around time, thus making it an ideal candidate for a Three-Dimensional (3D) CFD based design tool.

Keywords: Scroll pump, compressor, reed valve, valve dynamics, CFD

1. INTRODUCTION

Scroll compressors are widely used in many industries, including refrigeration, air-conditioning and automobile (as superchargers). It is believed that scroll compressors have the advantages of high efficiency, together with lower noise and vibration levels. There are various losses in a scroll compressor that need to be examined to achieve better efficiency, of which the most significant are flow losses, especially losses at the discharge (Hirano et al., 1989). CFD simulations can be a useful tool in understanding the flow field and help to reduce the flow losses through design optimizations.

However, flow simulation of a scroll compressor has proven to be a challenging task. There are studies to solve it via simplified mathematical models (Wang et al., 2011), as well as Two-Dimensional (2D) simulations (Rovira et al., 2006). Truly unsteady, 3D numerical studies of the scroll compressor remain scarce (Feng et al., 2004 and Cui, 2006). The fluid volume of a scroll compressor consists of gas pockets with large deformation and micron-level leakage gaps. So mesh generation and movement is not a trivial task.

Moreover, check valves are often used at the discharge to prevent reverse flow. The dynamics of the check valve alone can have significant influence on the efficiency and noise level of the compressor. Due to its importance CFD simulations on the stand alone check valve can be found in many studies (Lee et al., 1998).

A more realistic and useful simulation, therefore, should incorporate both the compressor and the check valve as a system, rather than model them separately at the component level. Usually, setting up these simulations can be quite involved and require substantial CFD expertise, and thus not so accessible for compressor manufacturers.

The present work attempts to solve this problem. A new scroll compressor template is implemented into a template driven design tool, PumpLinx, and demonstrated on a generic scroll compressor. The template is capable of

generating a rotor mesh from imported CAD surfaces, and automatic remeshing it according to the rotor movement, with a simple and straightforward procedure. The rotor meshing and remeshing algorithms developed in this study are generic and can be used with any CFD software. Moreover, the check valve is also simulated with the available valve template, providing a complete and simple solution to the flow simulation of the scroll compressor.

2. SCROLL ROTOR TEMPLATE MESHER

To ensure mesh quality throughout the movement of the orbiting rotor, especially for areas with small gaps, a continuous high quality multi-block structured mesh is generated for the scroll compressor.

As the first step the rotor CAD surfaces (Figure 1) are imported into the software and the stationary and orbiting rotors are identified. These surfaces can be easily output from any CAD package as STL files. The mesh generator is designed in such a way that the meshes are independent of the initial relative positions of the stationary and orbiting rotors. To define the rotating motion some geometric parameters, such as the center of both rotors and the rotation axis, are also provided as inputs.

Figure 1: A typical CAD surfaces input by user, stationary rotor in grey, orbiting rotor in red.

To start the meshing process, a 2D outline is extracted from the imported 3D surfaces and identified in step 1. Then the template mesher will search for several key geometry points, through which several partition lines are drawn and the computational domain is divided into meshing zones with high quality shapes.
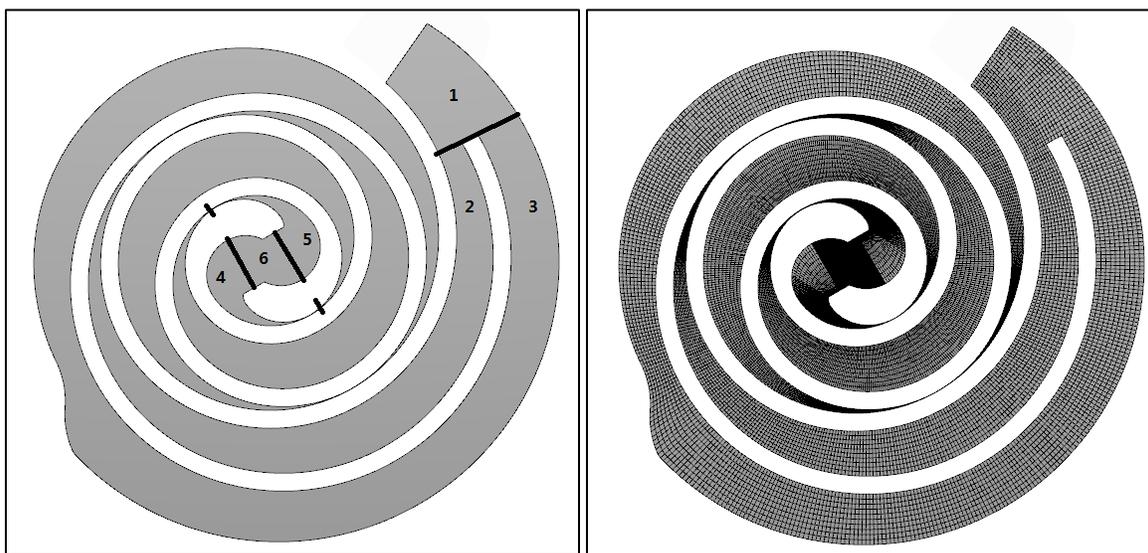
Figure 2: Rotor domain partitions (left) and resulting mesh (right)

Figure 2 shows that six partitions are created for the scroll rotor domain. Partitions 1 through 3 are the two branches of the spiral fluid volumes and the outer boundary area. These narrow, long fluid volumes are meshed by lines that are normal to either rotor outline, with evenly distributed nodes between the rotors.

The discharge area is further divided into Partitions 4-6. Partitions 4 and 5 are established in such a way that at most one "contact" point is allowed in each partition. For a continuous mesh, the solids are not allowed any actual contact with each other, but instead relatively small gaps are maintained. The template mesher will first identify the possible "contact" point, and uses lines that are normal to either solid wall to mesh the area close to the "contact", to ensure high level of orthogonality, while uniformly distributed nodes are used to draw mesh lines for the rest of the volume. Special treatments are also employed to preserve the possible sharp corners in these partitions.
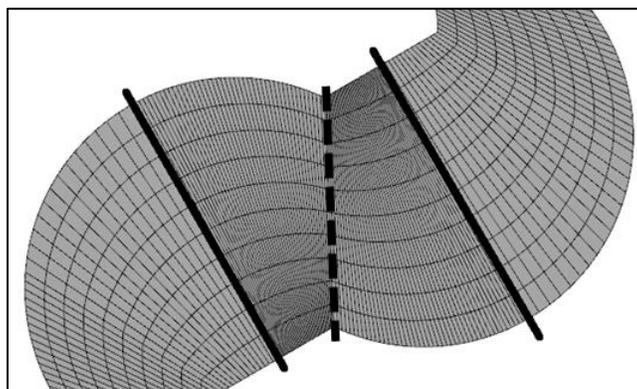
Figure 3: Further division of Partition 6.

In Partition 6, there could be as many as two "contact" points. For meshing purposes, the partition is further divided into two smaller ones. The dividing line is defined such that it splits the domain into two symmetric ones for symmetric rotors, and it has the smallest angle from normal direction on both rotor walls (shown as the center dash line in Figure 3). After the division, the same procedure as in Partitions 4 and 5 is applied to each of the two smaller partitions.

After the 2D mesh is created, it is stretched in the third direction to form a 3D grid. The meshing process is repeated for each time step when the orbiting rotor moves to a new position. Figure 4 shows the meshes for 4 different positions in one revolution.


a). 0 degrees	b). 90 degrees
c). 180 degrees	d). 270 degrees
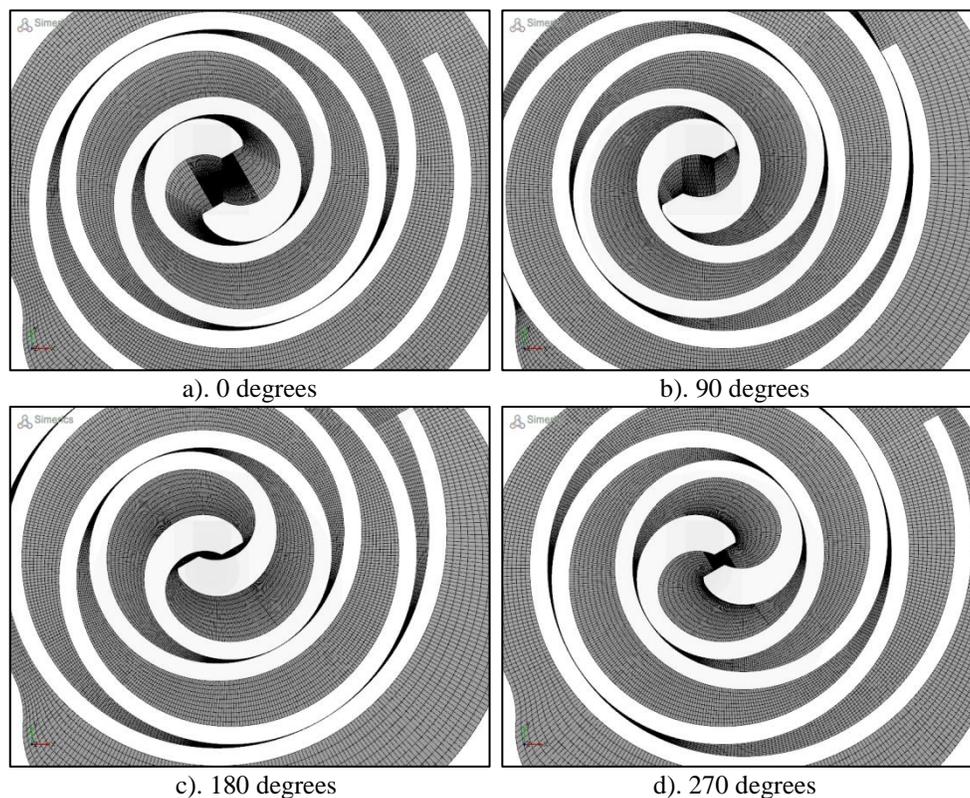
Figure 4: Scroll rotor meshes at different angles.

The new template mesher described here has been applied to various types of scroll rotors, with either smooth or abrupt outlines at the discharge, symmetric and non-symmetric rotor setups, etc. This new meshing procedure provides a simple way to generate high-quality mesh for the complicated fluid domain, requiring only CAD surface input and several user parameters.

3. REED VALVE SIMULATION

The reed valve is commonly used at the discharge of the scroll compressor to prevent reverse flow. It is typically made of a thin layer of metal with one end fixed, and the other end free. The valve will bend under fluid force from one direction to let the fluid flow through it (Figure 5). Due to strong interaction force and small inertia of the reed valve the coupling between scroll pump and reed valve becomes a very stiff Fluid Structure Iteration (FSI) system to solve. A semi-implicit proprietary procedure is applied to eliminate the requirement of extremely small time steps of explicit coupling methods.
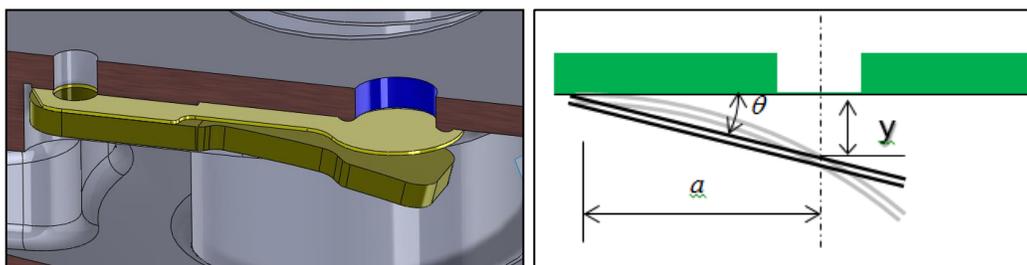
Figure 5: Geometry and schematic representation of a reed valve.

In the current study, however, the reed valve is modeled as a flip valve, or in other words, a thin plate of rigid-body rotation motion instead of a bending motion. The dynamic parameters, such as torsional spring constant, moment of inertia and preload torque, need to be chosen judiciously (Ding et al., 2014). This is a reasonable simplification, because for a reed valve that covers a small port opening, we can assume it to be a cantilever beam with a concentrated load at the center of the opening. In this case, the deflection at the opening is given by

$$y = \frac{Pa^3}{3EI} \qquad (1)$$

where P is the load, a is the length from fixed end to the center of the opening, E is the modulus of elasticity, and I is the area moment of inertia. From Figure 5, we can see that $\theta \approx y/a$, and the torque from the load can be computed as $T = Pa$, therefore, we have

$$T = \frac{3EI}{a}\theta \qquad (2)$$

As is shown above, the rotation angle at the opening center has a linear relation to the torque applied. Therefore, with carefully chosen dynamic parameters of the flip valve, the port opening center can have the same deflection as a bending valve under the same load.

The mesh of the flip valve is generated by an available template mesher from PumpLinx. Figure 6 shows the geometry and mesh of the reed valve.
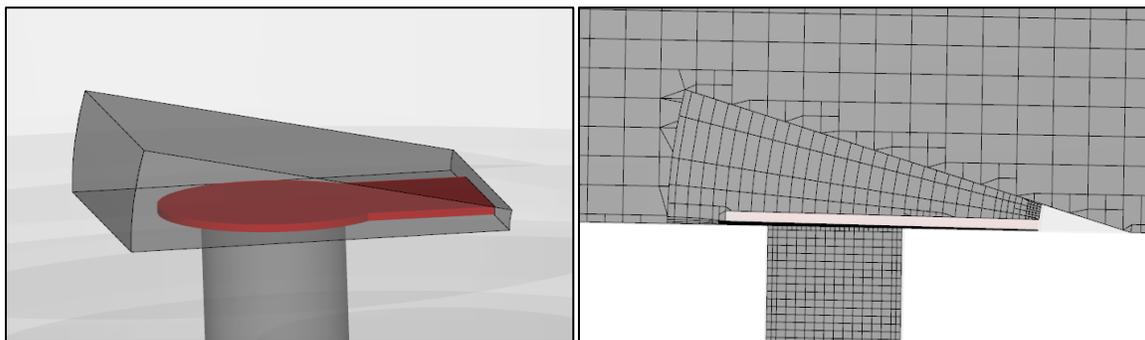

Figure 6: Reed Valve Geometry and Mesh.

4. A CASE STUDY

A generic scroll compressor at 5,000 RPM (Figure 7) was chosen to test and demonstrate the algorithm developed in this study. After meshing the scroll rotor and the reed valve using the above templates, a binary-tree based unstructured mesh is generated for the inlet and outlet volumes. These separate volumes are connected via "Mismatched Grid Interfaces" to form a continuous domain, consisting of 248,824 cells.

PumpLinx is widely used in device simulations with moving/rotating parts (Jiang et al., 2008 and Kovacevic et al., 2014). It uses a proprietary SIMPLES algorithm to solve the Navier-Stokes equations coupled with the energy

equation, the ideal gas law and the k-ε model. In addition, a built-in dynamic solver together with a built-in valve template is applied to solve the reed valve motion after the integrated torque is computed at each time step.

Fixed static pressure boundaries are prescribed at both inlet (1 bar) and outlet (4.5 bar). The simulation is performed on a system with quad-core Intel Xeon W3520 CPU at 2.67GHz. It took about 1 hour and 50 minutes for one scroll revolution, with three revolutions needed to reach a periodic state.
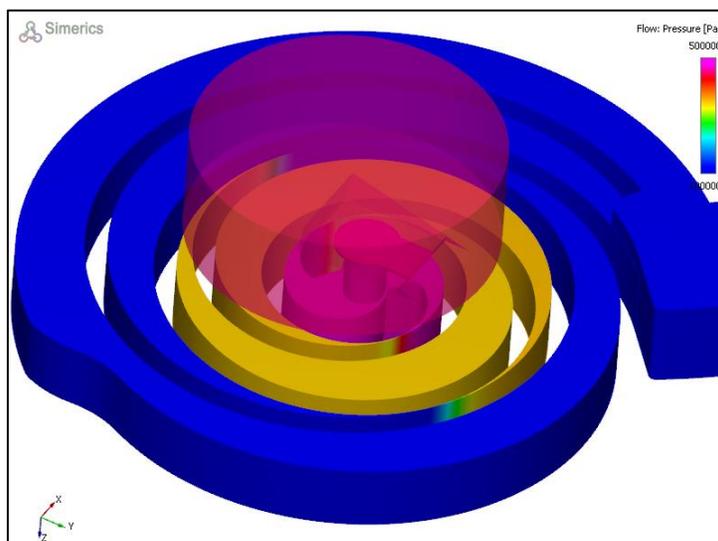

Figure 7: Instantaneous pressure distribution of the fluid domain.

The instantaneous pressure field shown in Figure 7 clearly demonstrates the three compression pressure levels of the scroll compressor. Figure 8 shows the volumetric flow rate at inlet and outlet. The valve displacement angle is shown in Figure 9. These results show that, for this specific design, the reed valve undergoes several oscillation cycles during one revolution, but is quite effective in preventing reverse flow.
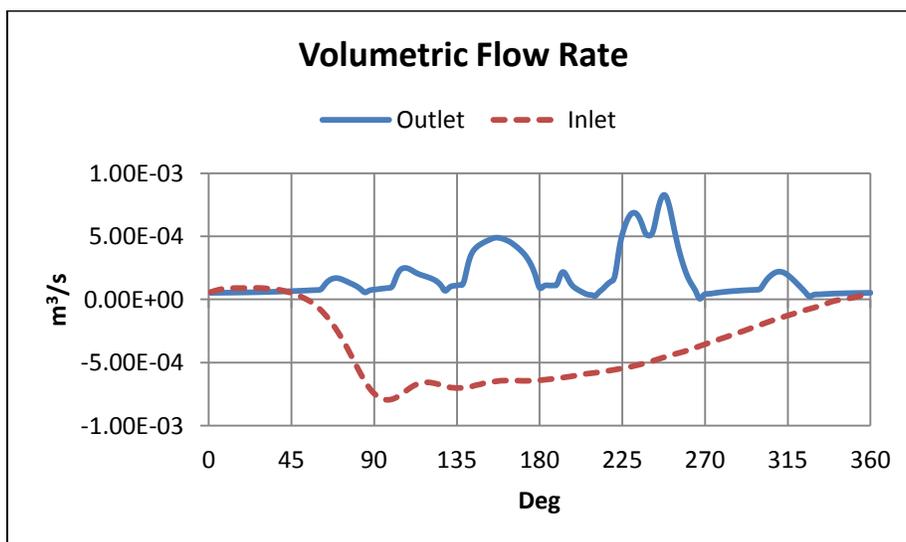
Figure 8: Volumetric flow rate at the inlet and outlet.
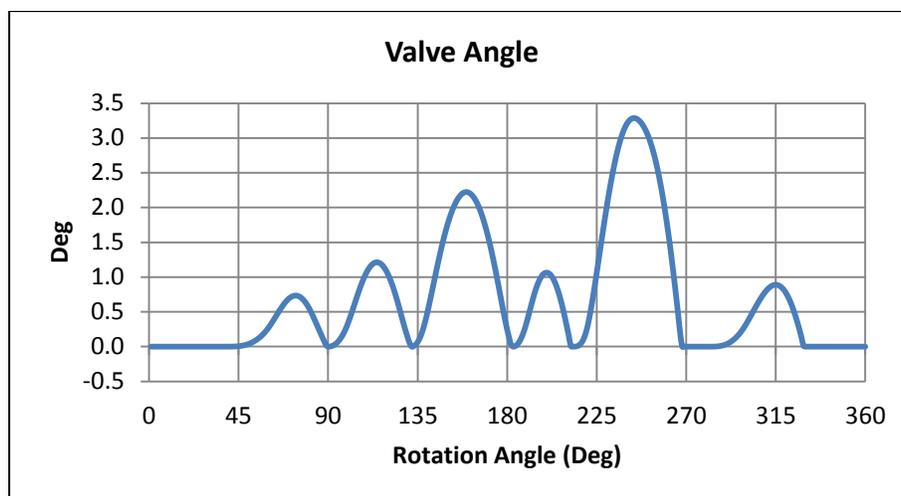
Figure 9: Reed valve movement during one revolution.

5. CONCLUSIONS

A new template mesher that can streamline the pre-processing of the scroll rotor is developed and applied to a generic scroll rotor. A reed valve that prevents the reverse flow is also included at the discharge port of the model.

With easy setup and a very efficient numerical solver, the unsteady flow field of scroll compressor can be simulated within a day. The results of the simulation provide insightful flow information to guide the optimization and design of such systems, including pressure/velocity distribution, flow rate and the angular displacement of the dynamic valve.

The ease of use and the short simulation turn-around time makes the proposed method an ideal candidate for simulations of scroll compressor systems.

REFERENCES

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

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

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.

Hirano. T., et al., "Development of High Efficiency Scroll Compressor for Heat Pump Air Conditioners", 1989, *Mitsubishi Heavy Industries, Ltd., Tech. Rev*. Vol. 26, No. 3, p: 512-519.

Jiang Y., Furmanczyk M., Lowry S., Zhang D., and Perng C.Y., 2007, A Three-Dimensional Design Tool for Crescent Oil Pumps. SAE p. 2008-01-0003.

Kovacevic, A., Rane, S., Stosic, N., Jiang, Y., Furmanczyk, M. and Lowry, S., "Influence of approaches in CFD Solvers on Performance Prediction in Screw Compressors", 2014, 22nd International Compressor Engineering Conference, Paper 1124.

Lee, J.K., Jung, K., Lee, J.W., "Numerical Analysis on Unsteady Flow in Valve. Chamber of Scroll Compressor", 1998, *Theses Collection, Kyung Hee Univ.*, Seoul, Korea 27 I 403~409.

PumpLinx 3.4, 2014, User help manual.

Rovira, J., Rigola, Joaquim, Oliva, Asseni, Perez-Segarra and Carlos D., "Numerical Simulation and Experimental Comparison of Refrigeration and Air Conditioning Scroll Compressors. Application to Trans-Critical Carbon Dioxide Cycles" (2006). *International Compressor Engineering Conference*. Paper 1801.

Wang, J., Yang L., Luo, X., Mangan, S.; Derby, J.W., "Mathematical Modeling Study of Scroll Air Motors and Energy Efficiency Analysis—Part I,", 2011, *Mechatronics, IEEE/ASME Transactions*, vol.16, no.1, p.112-121.