

2016

# Numerical Simulation of the Flow Inside a Scroll Compressor Equipped with Intermediate Discharge Valves

Alain Picavet

*Danfoss Commercial Compressors, France, a.picavet@danfoss.com*

Brian Angel

*Danfoss Commercial Compressors, France, brian.angel@renuda.com*

Follow this and additional works at: <https://docs.lib.purdue.edu/icec>

---

Picavet, Alain and Angel, Brian, "Numerical Simulation of the Flow Inside a Scroll Compressor Equipped with Intermediate Discharge Valves" (2016). *International Compressor Engineering Conference*. Paper 2421.

<https://docs.lib.purdue.edu/icec/2421>

This document has been made available through Purdue e-Pubs, a service of the Purdue University Libraries. Please contact [epubs@purdue.edu](mailto:epubs@purdue.edu) for additional information.

Complete proceedings may be acquired in print and on CD-ROM directly from the Ray W. Herrick Laboratories at <https://engineering.purdue.edu/Herrick/Events/orderlit.html>

# Numerical simulation of the flow inside a scroll compressor equipped with Intermediate Discharge Valves

Alain PICALET, Brian ANGEL

Danfoss Commercial Compressors, 01600 Trevoux ; France

Renuda, Belleville sur Saône ; France

[a.picavet@danfoss.com](mailto:a.picavet@danfoss.com)

[brian.angel@renuda.com](mailto:brian.angel@renuda.com)

## ABSTRACT

This paper presents the results of CFD simulations of the compression process of a scroll compressor. The compressor geometry accounts for the final scroll gas pocket, two intermediate discharge valves (IDVs), the central discharge zone and the upper shell or high pressure (HP) zone. The numerical model uses a real gas equation of state to determine gas properties during the compression process and accounts for the motion of the orbiting scroll and the IDVs using a mesh smoothing and remeshing algorithm. The IDVs are represented as a spring mass system with their movement controlled via the pressure difference around the valve. Appropriate pressure based boundary conditions are used at entry to the third gas pocket and at the exit of the upper shell. An initial analysis of the results indicates that it is possible to achieve time accurate results of the pressure field throughout the flow domain and also determine the impact of the IDVs on scroll performance. The results will also enable a more thorough analysis of the fluid flow and compression process inside the scroll in order to improve its performance.

## 1 INTRODUCTION

The use of CFD for modelling scroll compressor fluid flows is now standard practice due to the technical capabilities of commercial CFD codes and the use of high performance machines. Typical modelling processes at Danfoss now account for the scroll compression process using an orbiting scroll, IDVs and flank and tip leakage.

Due to its significant experience in this area, Danfoss uses IDVs to maintain compressor performance when it is operating at low pressure ratios. At these pressure ratios, the pressure in the final gas pockets can be higher than that in the HP zone resulting in lower efficiency and excessive power consumption. The IDVs are used to bleed off this excessive pressure and, as previously mentioned, maintain performance.

Detailed experimental analysis (Giniès et al. and Picavet et al.) and CFD modelling of simplified scroll compressors (Angel et al.) have been previously presented. The work presented here focuses on the CFD modelling of a scroll compressor using two IDVs and the comparison of these results with experimental data.

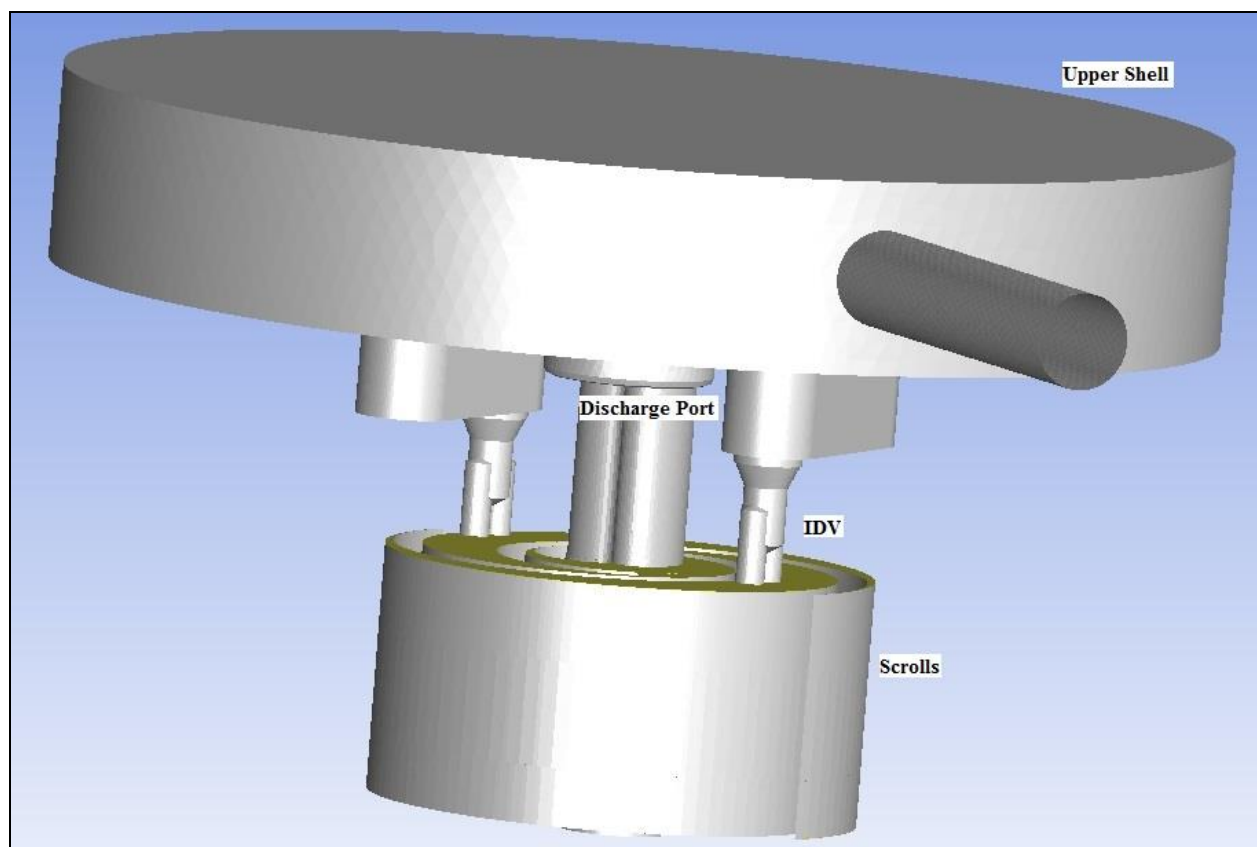
## 2 NUMERICAL MODEL

The current numerical model is based on the commercial CFD solver FLUENT. For all simulations, the flow is assumed to be unsteady, three-dimensional, compressible and turbulent with adiabatic walls. The gas is assumed to be a real gas, R410A, modelled using the Soave-Redlich-Kwong equation of state available in FLUENT. The initial properties of R410A to be used in Fluent are computed using REFPROP, the software developed by the NIST.

Turbulence is modelled using the  $k-\omega$  model. The boundary conditions used are pressure-inlet, pressure-outlet and wall. A second order scheme is used for the discretization of the equations. The SIMPLE algorithm with default relaxation is used to ensure the convergence of the solution in time, using a first order stepping method, with up to 30 iterations per time-step. FLUENT has been set up save flow field data 40 times per revolution (volume or mass averaged data, flow field results and various images on different viewing planes).

The computational domain is shown in Figure 1 and consists of the final gas pocket, the main discharge port, two IDVs and the HP zone of a 2.4 volume ratio machine, the cooling capacity of which being 25 tons. The numerical model uses the mesh motion algorithm and appropriate UDFs to control the movement of the orbiting scroll and the movement of the IDVs. Mesh quality is maintained using the mesh smoothing and remeshing algorithms. The simulations are run using a time step size designed to ensure that the remeshing process does not generate negative volume cells. The initial grids of the pockets of the scrolls are shown in figure 2.

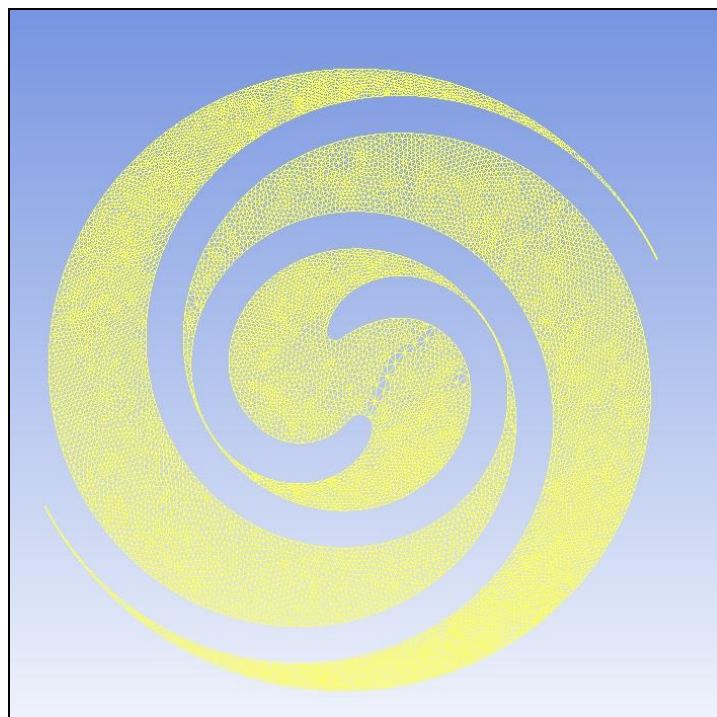
The scroll compressor is running with the refrigerant gas R410A at 2900 rpm with an operating point characterized by an evaporating temperature of 10 °C and a condensing temperature of 40 °C with a superheat of 10 °C. The pressure ratio at this point is 2.23, which ensures that the IDVs will open for each revolution of the orbiting scroll.



**Figure 1:** Computational domain: scrolls, discharge port, IDVs and upper shell (HP zone)

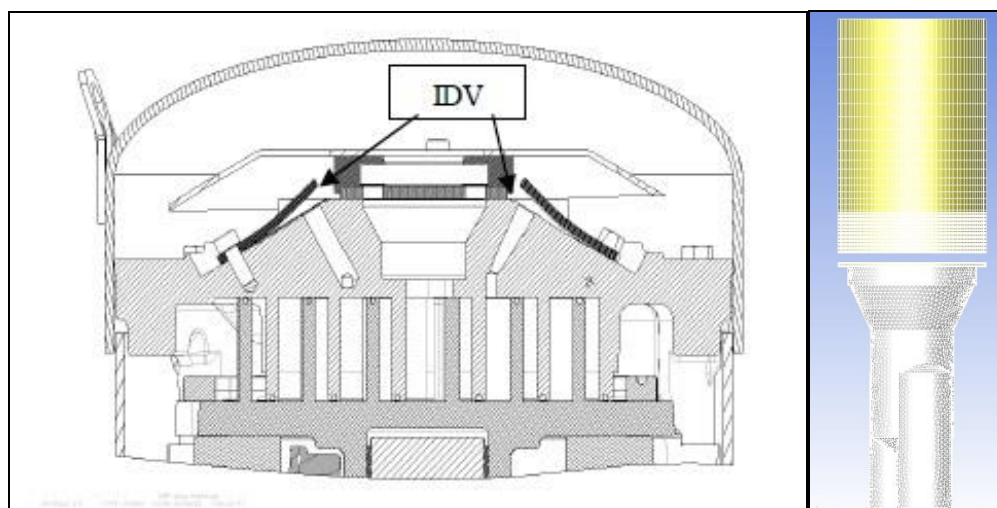
Danfoss compressors are equipped with tip seals located on the top of the static and orbiting scroll spirals. The central part of each spiral interacts with the discharge port hence there is no tip seal present due to the risk of the seal being pulled out of its housing. Where present, the seal is assumed to cover the entire width of the spiral so this zone is not meshed. Where the seal is not present in the central zone of each spiral (hence the gas can flow through this tip gap) the tip gap zone is meshed with at least 5 layers of cells for gradient capture. The tip gap mesh does not deform but it rotates with the orbiting scroll.

Flank leakage is controlled via the use of appropriate UDFs which limit the mass flow through flank gap to appropriate flow rates.



**Figure 2:** Gas pocket initial mesh

Figure 3 presents a sectioned view of a scroll with typical IDV. For the numerical model, the geometry of the Intermediate Discharge Port is respected, however the valve has been simplified to a solid round disc which is subject to the gas pressure in the pocket and the upper shell. The disc movement is vertical and is modeled as a spring-mass system. The stiffness and the mass of the real valves are used to model the movement. When the valve is closed, a tiny gap exists between the lower surface of the valve (or disc) and its housing. This gap is meshed and the cells in this zone are assigned viscosity of the order of  $100\text{Pa}\cdot\text{s}$  when the valve is in its closed position. This high viscosity ensures a negligible leakage flowrate through this zone. When the valve starts to open, the viscosity is reset to its normal value and the layers of cells are “inflated” via a UDF which is controlled by the dynamic mesh algorithm.



**Figure 3:** Comparison between real and modelled IDVs

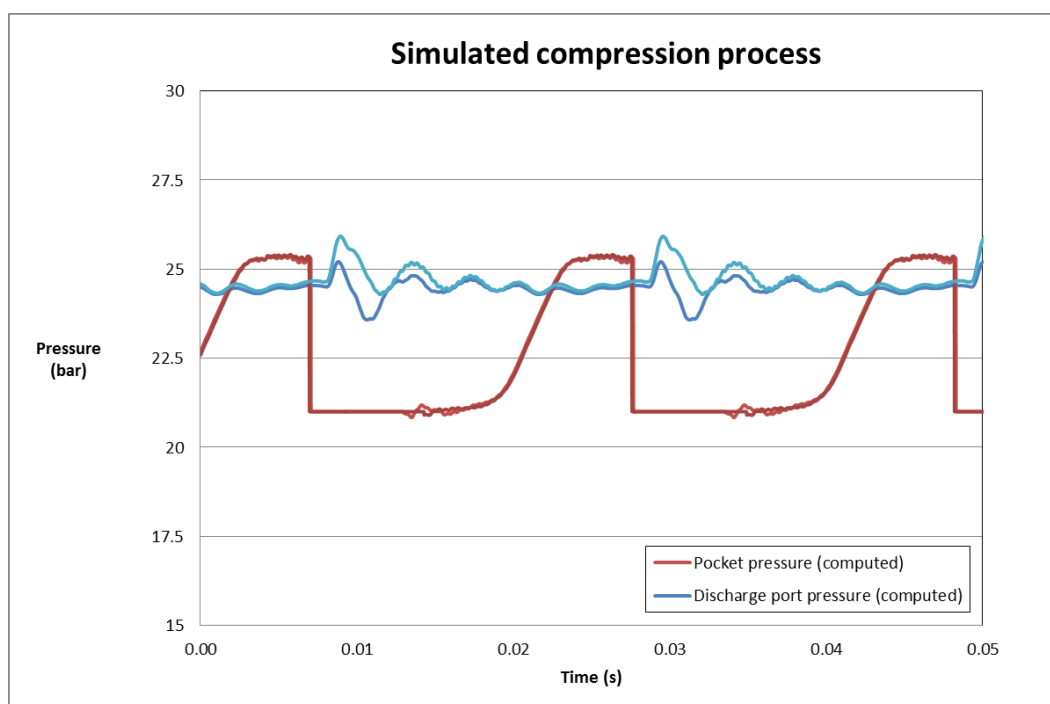
### 3 RESULTS AND DISCUSSION

#### 3.1 Analysis of pressure curves

Figure 4 presents the pressure in the final gas pockets (red curves) and the discharge port (blue curves) at different monitoring points. Given that only the final gas pockets are modelled, the predicted pressure is only “measured” for a given part of the compression cycle. The initial pressure used at the inlet to each of the final gas pockets is set to the pressure at the end of the compression of the intermediate or second gas pockets. These inlet pressures are calculated using 1D software developed and validated by Danfoss.

Figure 4 shows the pressure increasing as the pocket volume reduces then stabilize as the IDVs open due to the pressure difference between the gas pockets and the HP zone. The discontinuous reduction in pocket pressure is due to the monitoring point moving outside of the flow domain.

In the main discharge port, the pressure can be seen to be oscillatory which is linked to the opening of the scrolls at the beginning of discharge process. This opening means contact between three volumes at different pressures and results in rapid mass transfers of gas and highly turbulent flow between the gas pockets and the discharge port.



**Figure 4:** Distribution of pressure in the pockets and the discharge port

Figure 5 compares the pressure distribution and IDV motion measured during a previous test campaign (on a machine with the same volume ratio running at the same operating point) and compares this data with the CFD results. The tests show that the valve starts to move when the pressure difference between the pocket and the discharge port reaches more than 1.5 bar, resulting in a time shift of 3 milliseconds. The simulation shows a pressure difference of less than 0.5 bar before the beginning of IDV motion, with a time shift of 0.5 millisecond and a longer opening duration. The CFD also overestimates the discharge flowrate through the IDV resulting in an underestimated maximum pressure in the pocket. Hence, the CFD model should be tuned to obtain more reliable results.

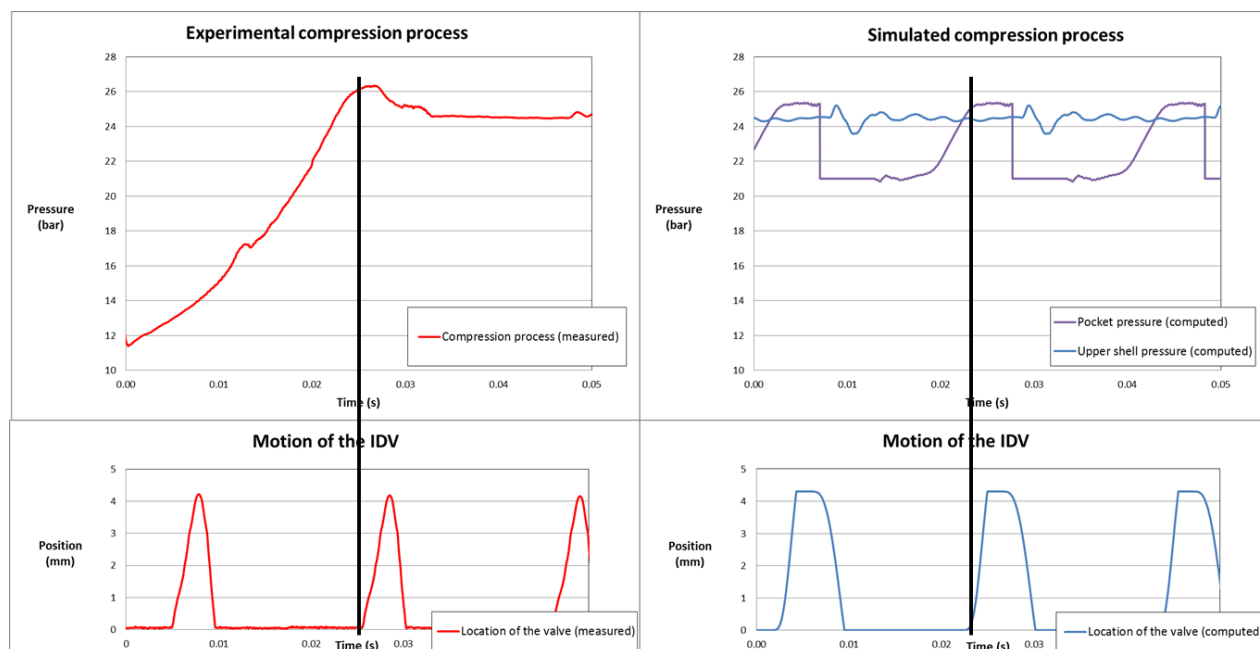


Figure 5: Comparisons of experimental and computed compression process

### 3.2 Analysis of the flow field

Figure 6 presents the velocity magnitude in the scroll compressor on different viewing planes. Here, the angle  $0^\circ$  corresponds to the beginning of the compression process in the final gas pockets. At  $63^\circ$ , the IDVs start to open and at  $144^\circ$ , they are closed. At  $162^\circ$ , the spirals open onto the discharge port.

The gas velocity can be seen to be in excess of 30m/s in the IDV ducting as the IDVs open and in the discharge port as the gas pockets open onto this zone. The gas velocity in the tip gap can also exceed 30m/s. An analysis of the flow indicates that it is highly turbulent with numerous vortices being generated in the gas pockets and in the discharge port, explaining (at least in part) the oscillations seen in Figure 4. At  $144^\circ$ , high gas velocities are seen in the IDV ducting and this is due to the gas flowing back into the gas pockets due to the pressure in the HP zone being higher than that in the gas pockets.

The analysis of the flow in the IDV ducts has shown that the flow is not identical in each duct due to differences seen in each gas pocket but also locally around each valve in the HP zone. This flow asymmetry is of interest and will be investigated further.

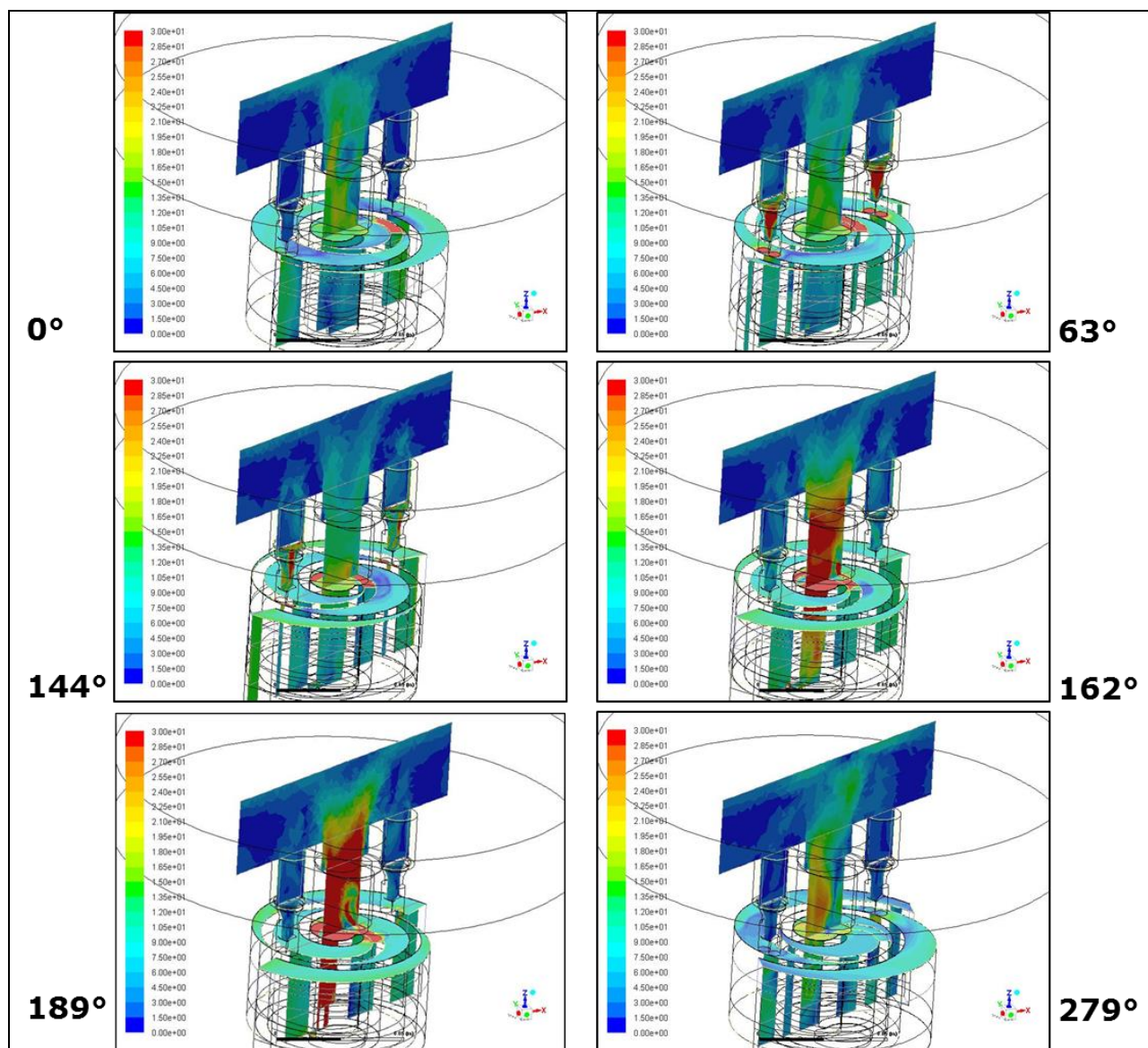


Figure 6: Velocity magnitude contours at different angles of rotation

Figure 7 presents the velocity field around the IDVs and in the discharge port as the spirals open. In the IDV ducting, it can be seen that the gas velocity is of the order of 50m/s and that there is a large zone of recirculating gas downstream of the valve when it is fully open. This will contribute to the overall losses of the scroll compressor hence a possible redesign of the shape of the IDV valve to reduce this loss could be considered (although it should be understood that these losses are only for approximately 1/6<sup>th</sup> of a complete revolution).

As the spirals open onto the main discharge port, the gas can be seen to be flowing into the port at velocities exceeding 50m/s resulting in highly turbulent flow with large recirculating zones. This will also contribute to the overall losses of the compressor and indicates that the design of the discharge port could be reconsidered in order to improve performance.

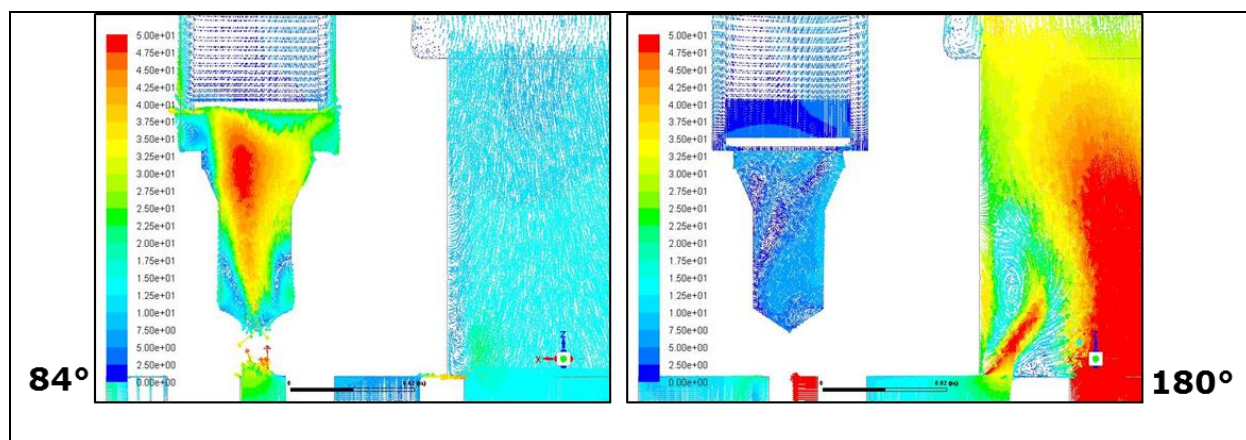


Figure 7: Velocity vectors around the IDV and at the edge of the discharge port

Figure 8 presents the vorticity calculated by FLUENT where there is a significant increase in vorticity as the discharge process starts. This confirms the analysis of the flow field presented in Figures 6 and 7 and indicates that CFD can be used with success to analyse scroll performance and assist in the design process.

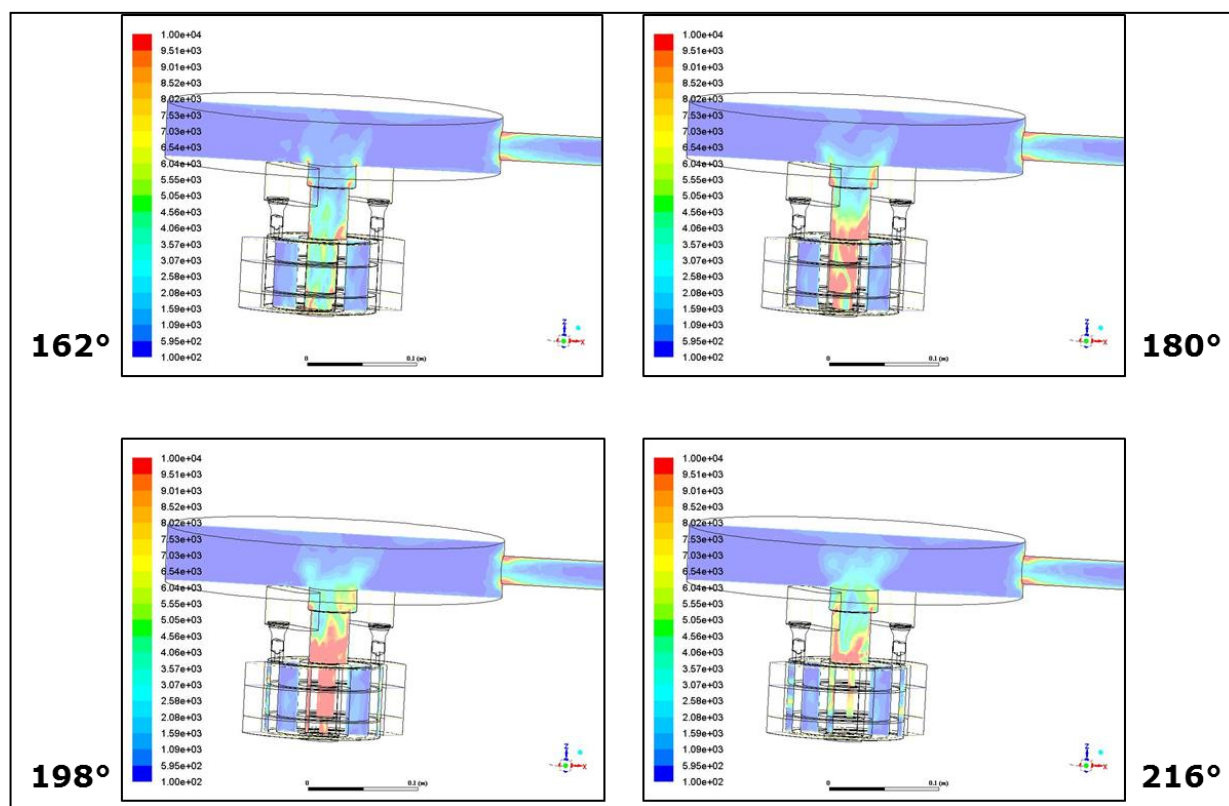


Figure 8: Vorticity contours at discharge

## 4 CONCLUSIONS

The work presented here has shown that it is possible to simulate, with an acceptable level of accuracy and CPU time, the flow in a scroll compressor whilst accounting for important details such as tip and flank leakage and intermediate discharge valves.. These results of such simulations provide essential details concerning the flow field and how the different components of the compressor interact with one another. These results are also extremely



useful to gain an understanding of the unsteady physical phenomena occurring during the compression process, especially in the discharge port, and how it would be possible to improve compressor performance.

## REFERENCES

P. Giniès, C. Ancel, D. Gross : Scroll Compressors and Intermediate Valve Ports. *7<sup>th</sup> International Conference on Compressors and their Systems*, London, 5-6<sup>th</sup> September 2011.

B. Angel, P. Giniès, D. Gross, C. Ancel : CFD modelling of scroll compressor intermediate discharge ports. *8<sup>th</sup> International Conference on Compressors and their Systems*, London, 9-10 September 2013.

Picavet, Alain and Giniès, Pierre : Experimental Pressure-Volume Diagrams of Scroll Compressors. *22<sup>nd</sup> International Compressor Engineering at Purdue*, 14-17<sup>th</sup> July 2014.