16th International Research/Expert Conference
”Trends in the Development of Machinery and Associated Technology”
TMT 2012, Dubai, UAE, 10-12 September 2012

PERFORMANCE OPTIMIZATION OF SCREW COMPRESSORS BASED ON NUMERICAL INVESTIGATION OF THE FLOW BEHAVIOR BASED ON DIFFERENT GRID GENERATION APPROACH

Ahmed Kovacevic,
Sham Rane,
City University,
London, UK
A.Kovacevic@city.ac.uk

Maria Pascu
Howden Compressors Ltd,
Glasgow, UK
Maria.Pascu@howden.com

ABSTRACT

In the field of screw compressors, although the basic operation of flow systems is well known and the analytical methods for their performance prediction are well established, only few attempts of investigating the flow by means of CFD can be identified in the available literature.

In this paper, two important aspects of numerical investigation of the flow behavior are presented. Firstly, methodology for grid generation of constant pitch twin screw machines is available through SCORG© but it is currently not suitable for different topologies like that of a single screw or variable pitch rotors. Hence a currently available technology of grid remeshing has been used to demonstrate the CFD simulation of an adiabatic compression-expansion process in a reciprocating piston cylinder arrangement. This simulation gives a basis for application of this grid generation technique for screw compressors. The methodology tested for this paper uses a technique called key-frame re-meshing in order to supply pre-generated grids to the CFD solver as the solution progresses. It was concluded that customized tools for generation of CFD grids for such complex screw machines still remain to be developed.

Secondly, it is intended in this paper to put an emphasis on the current features and potential for development of SCORG© - a customized grid generator, to establish CFD as permanent tool for compressor design and optimization in the New Product Development process. The definition of an optimum meshing procedure and, subsequently, of appropriate boundary conditions, allowed the successful setup of flow simulations in an oil-free screw compressor and the prediction of the compressor performance, with a good degree of confidence. The flow analysis included both qualitative and quantitative evaluations of parameters like the variation of the pressure and temperature in the discharge chamber, torque, power, and entropy variation.

This detailed analysis allowed for design improvements of the discharge chamber to be implemented and “virtually” tested in order to determine an optimized compressor geometry, which will be subsequently incorporated in the actual modular casing of the compressor.

Keywords: CFD, Screw Compressor, Performance Optimization, Key-Frame, Re-meshing.

1. INTRODUCTION

Nowadays, the manufacturing techniques have become so advanced that the screw compressor rotor manufacturing can be done to very tight tolerances and the interlobe clearances created by the rotor meshing is in the order of few microns. There are many advantages in considering CFD as integrated part of the design and optimization process of screw compressors (SC). Probably the most noticeable efforts in the field of numerical analysis of SC were made by Kovacevic et. al. [2–6], where in addition to establishing a mesh procedure specific to such flow machines, the author also explains adequate boundary calculations to encourage good convergence and minimal numerical errors.

Similar efforts were made by Sauls and Branch [7], where the commercial code ANSYS-CFX was used for the detailed analysis of a refrigeration SC designed for use with R134a in air- and water-cooled chillers. Also benefiting from the mesh technique documented in [2–6], Steinmann [8] reported results from the modeling of a helical-lobed pump and a SC using ANSYS-CFX.

The methodology adopted for here uses a technique called key-frame re-meshing to supply pre-generated grids to the CFD solver as the solution progresses. To test the applicability of this methodology as a generic tool for screw compressors, an adiabatic compression-expansion process in a reciprocating piston cylinder arrangement was calculated and compared with diffusion equation based mesh smoothing w.r.t accuracy of results obtained with both the techniques.

The paper also attempts at contributing to the field by establishing CFD as a design tool for the optimization of SC during the New Product Development process at Howden Compressors Ltd. The CFD analysis focused on two main objectives. First, a baseline model for the optimization exercise was determined: the discharge chamber (discharge port, pipe and flanges) of the modular casing of a single-wall/ rolling-element bearing/ compressor, characterized by a rotor diameter of 127 mm. Second, the performance of the baseline was assessed against that of several new designs which aimed at improving the gas flow in the discharge chamber. The results of this analysis will be presented next.

2. USE OF GENERIC GRID GENERATION TO SIMULATE A SIMPLE PISTON CYLINDER CONFIGURATION

The purpose of this analysis is to compare the results from Diffusion Smoothing based Mesh Motion and Key-Frame based remeshing method with the theoretical results. It is expected that Diffusion Smoothing will be exact as there are no interpolation errors from one time step to the other. The theoretical process can be modeled by polytropic equation relating the gas pressure and volume. Since the process is adiabatic and reversible there will be no energy loses or gains in the control volume and the gas will return to its initial state. For the Key-Frame based remeshing a tetrahedral mesh is selected since in the real screw compressor the geometry is complex and hexahedral meshes are difficult to be generated.

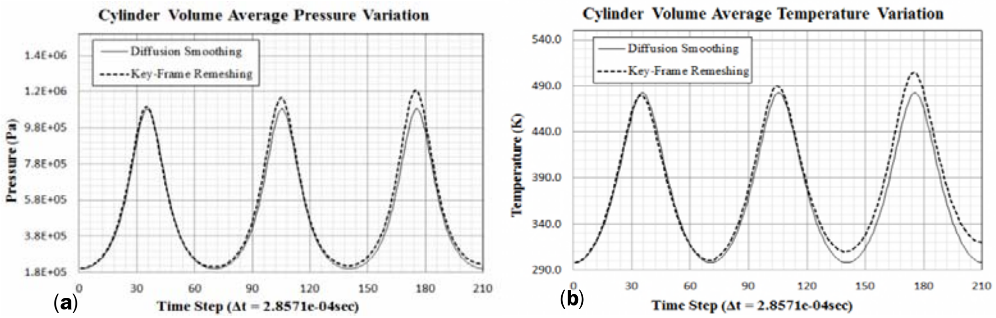

Figure 1. Comparison of (a) Pressure and (b) Temperature, with Diffusion Smoothing and Key-Frame remeshing

Figure 1a shows the change of pressure with time and Figure 1b shows the temperature change with time in the cylinder for both cases.

Table 1. % error in calculation of Pressure and Temperature

Quantity	% error					
	Ist Cycle		IInd Cycle		IIIrd Cycle	
	Case-1	Case-2	Case-1	Case-2	Case-1	Case-2
Absolute Pressure	-0.44	1.15	-0.42	5.38	-0.41	9.33
Static Temperature	-0.08	-0.53	-0.07	1.54	-0.05	4.57
Case-1 Diffusion Smoothing, **Case-2** Key-Frame Remeshing						

Table *1* shows the percentage error in the prediction of pressure and temperature by both methods and also its variation over multiple consecutive cycles. These results show that diffusion smoothing based method is highly accurate and conforms to the theoretically expected results for a deforming boundary formulation. But there are errors in the pressure and temperature prediction using the key-frame remeshing based method where the mesh is replaced after every time step. These errors can be attributed to the interpolation of results from one time step to the other on the replaced mesh. There could also be some violation of space conservation equation happening as the mesh is replaced every time step. As pointed out in [1] this can lead to artificial mass source errors in the continuity equation that can also accumulate with flow time. Limiting the mesh to be replaced only when cell quality goes bad should help is reducing the error but in case of complex topologies like the screw compressors this is very difficult. This analysis gave important information about the level of accuracy to be expected with the key-frame remeshing approach.

3. PERFORMANCE OPTIMIZATION OF TWIN SCREW COMPRESSOR USING SCORG© GRID GENERATOR

The baseline design for this optimization exercise was the discharge chamber (port, pipe and flanges) of the modular casing of a single-wall/ rolling-element bearing/ compressor, characterized by a rotor diameter of 127 mm. Several geometrical modifications were applied to the baseline, consisting of:

a. Elimination of the area contraction in the discharge pipe, as well as any sharp corners in the flow domain.
b. Several values of the diameter of the discharge flanges were analysed, i.e. 50, 60, 70 and 80mm.
c. Another check point for the new designs was the area progression of the flow from the discharge port and into the chamber, i.e. contractions should be avoided. The constraint was $A_{port/2} \le A_{pipe/2}$ and this was satisfied by all the proposed designs.

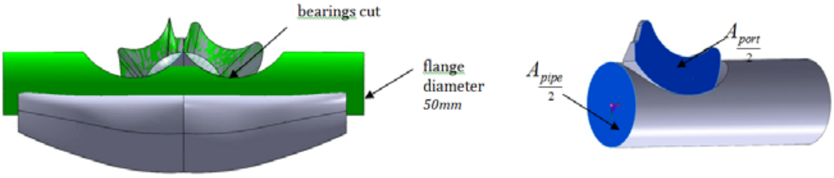

Figure 2. Modified discharge chamber (green) vs. the baseline and Area progression from the discharge port into the chamber

Generating the grids for rotor domains is by far the most challenging part of the entire meshing procedure, as both micro- and macro- scales elements have to be solved (the interlobe clearances is in the order of several microns, whilst the rest of the rotor body measures 200 mm). In this case, a technique dedicated to screw compressor rotors was employed, as described by Kovacevic [3], included in SCORG (**S**crew **C**ompressor **R**otor **G**eometry grid generator).

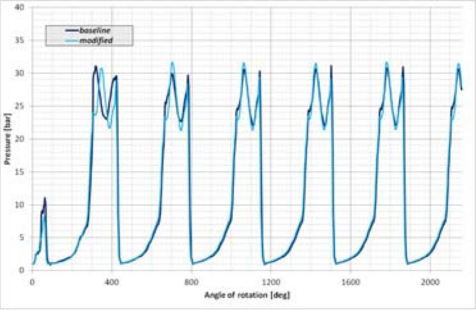

Figure 3. Pressure variation at the rotor interface in the discharge chamber

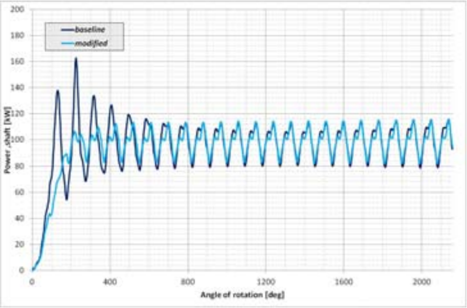

Figure 4. Power required by the compressor shaft

A first (qualitative) evaluation of the numerical results can be carried out based on representative pressure and temperature plots, which prove helpful in observing the compression cycle, from the compressor start-up (warm-up process) until the end of the cycle. In *Figure 3* the pressure variation in the discharge chamber, at the interface with the two rotors, is depicted. The compressors were compared, baseline and the modified design, and it was observed that they are characterized by similar behavior over the six full rotations.

Figure 4 depicts a similar evolution of the shaft power along six male rotor rotations. The similar behavior of the two compressors is due to the fact that, as positive displacement machines, screw compressors consume power mostly due to increase in internal energy of the fluid and only marginal difference may occur due to flow losses in ports which is associated with the kinetic energy of the flow. In order to compare the performance of the proposed models, the flow rate in the discharge chamber was measured for both models. It was observed from these calculations that the modified design is characterized by a slightly smaller flow (approx. 14% less) which means that smaller losses occur in this chamber when compared to the baseline model.

4. CONCLUSION

The main objective of the present paper was to demonstrate the capability of the established CFD procedure by use of SCORG and ANSYS-CFX to optimize geometry of the discharge port.

Simultaneously, a methodology of key frame re-meshing was tested but all attempts to apply this generic method to solve flow within twin screw compressor failed due to complexity of numerical mesh. This study showed that though it is possible to simulate some of the complex configurations of screw compressors by using general purpose grid generators, there are a many limitations like: Time consumption for pre-processing, lack of accuracy, inability to include leakages, and limitations in complex domains such as that of a twin screw compressors. Therefore the need exists to develop customized tools to generate CFD grids for complex screw machines such as single screw, variable pitch machines, etc.

The CFD analysis using SCORG followed two venues: firstly, a baseline model for the optimization exercise was determined, i.e. the discharge chamber of a modular casing of a single-wall/ rolling-element bearing/ compressor; secondly, the performance of the baseline was assessed against that of several new designs which aimed at improving the gas flow in the discharge chamber.

After bringing several geometrical improvements to the baseline model in order to stabilize the flow in the discharge chamber and reduce the turbulence, a thorough flow analysis, which included pressure plots and the definition of various performance indicators, was carried out. It was observed that the geometrical modifications of the discharge chamber produced the desired effect and turbulences in the discharge domain were reduced. Additionally, when comparing the performance of the various compressor models, it was observed that the modified design was characterized by smaller flow losses in the discharge chamber when compared to the baseline model.

Such numerical analysis will allow for design improvements of the discharge chamber to be implemented and subsequently tested on the adequate test rig and incorporated in the actual casing of the compressor.

5. REFERENCES

[1] Ferziger J. H. and Peric M., 1996, Computational Methods for Fluid Dynamics, ISBN 978-3-540-42074-3, Springer, Berlin, Germany.

[2] Stosic, N., Smith, I., and Kovacevic, A., “Screw compressors. Mathematical modeling and performance calculation”, Springer Verlag, 2005

[3] Kovacevic, A., Stosic, N., and Smith, I., “Screw compressors. Three dimensional computational fluid dynamics and solid fluid interaction”, Springer Verlag, 2007

[4] Kovacevic, A., Stosic, N., and Smith, I., “Three-dimensional numerical analysis of screw compressor performance”, Journal of Computational Methods in Sciences of Engineering, vol.3, No.2, 2003

[5] Kovacevic, A., Stosic, N., and Smith, I., “Analysis of screw compressor by means of three-dimensional numerical modeling”, International conference on compressor and their system, IMechE Conference transactions 2001-7, London, 2001

[6] Mujic, E., Kovacevic, A., Stosic, N., and Smith, I., “The influence of port shape on gas pulsations in a screw compressor discharge chamber”, Centre for positive displacement compressor technology, City University, London

[7] Sauls, J., and Branch, S., “CFD analysis of refrigeration screw compressors”, Ingersoll Rand, 2009

[8] Steinmann, A., “Numerical simulation of fluid flow in screw machines with moving mesh techniques in ANSYS CFX”, Schraubenmaschinen 2006; VDI Verlag GmbH, 2006