View metadata, citation and similar papers at core.ac.uk
brought to you by CORE
provided by Purdue E-Pubs

Purdue University
Purdue e-Pubs

International Compressor Engineering Conference | School of Mechanical Engineering

2014

Numerical Investigation Of The Leakage Flows In Twin Screw Compressor Rotors

Maria Pascu
Howden Compressors Ltd, United Kingdom, maria.pascu@howden.com

David Buckney
Howden Compressors Ltd, United Kingdom, david.buckney@howden.com

Manoj Heiyanthuduwage
Howden Compressors Ltd, United Kingdom, manoj.HEIYANTUDUWAGE@howden.com

Graeme Cook
Howden Compressors Ltd, United Kingdom, graeme.cook@howden.com

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

Pascu, Maria; Buckney, David; Heiyanthuduwage, Manoj; and Cook, Graeme, "Numerical Investigation Of The Leakage Flows In Twin Screw Compressor Rotors" (2014). *International Compressor Engineering Conference.* Paper 2321.
https://docs.lib.purdue.edu/icec/2321

This document has been made available through Purdue e-Pubs, a service of the Purdue University Libraries. Please contact 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 Analysis of the Influence of Leakage Flows on Screw Compressors Performance Prediction

Maria PASCU*, David BUCKNEY, Manoj HEIYANTUDUWAGE, Graeme COOK
Howden Compressors Ltd,
Glasgow, UK

Maria.Pascu@howden.com
Tel: +44 (0) 141 885 7720
Fax: +44 (0) 141 303 5223

* Corresponding Author

ABSTRACT

Screw compressors are complex flow systems, but operate upon simple considerations: they are positive displacement machines consisting of meshing rotors contained in a casing to form a working chamber, whose volume depends only on the angle of rotation. Although the basic operation of twin screw compressors is well understood and the analytical methods for performance prediction are well established, the CFD analysis of such machines is still in its early days and requires more research in order to mature to the level of standardised application, as is the case with most turbomachinery applications.

The performance of screw compressors is highly affected by leakages, which are dependent on various clearances and the pressure differences across these clearances. The numerical simulation of the leakage flows is the challenging aspect of running CFD in screw compressors, as the flow domain reduces in tip areas to the order of several microns. Commercial grid generators struggle in reproducing the flow path with good quality elements and user defined meshing techniques are currently available to allow the remeshing of the rotors domain to capture the modified aspect of the compression chamber at the start of each time step. This technique allows the full compressor performance prediction by means of CFD but does not allow for local mesh refinement in the tip area, where the turbulent character of the flow requires adequate boundary layer considerations. This is the area of concern in the present paper, where several means of investigating the flow in the very tight tip clearance area are proposed: 1) a three-dimensional model of all interacting compression chambers to investigate the effects of employing mesh-deforming techniques for the compressor simulation; 2) a simplified two-dimensional leakage model to identify the mesh refinement requirements for appropriate turbulence modelling.
Keywords: screw compressors, CFD, leakage flows, tip seal

1. INTRODUCTION

Although the basic operation of twin screw compressors is well known and the analytical methods for their performance prediction are well established, few attempts of investigating the flow in screw compressors by means of CFD can be identified in the available literature. Nevertheless, there are many advantages in considering CFD as an integrated part of the design and optimization process of screw compressors (SC). This is mostly because CFD complements the experimental and analytical efforts by providing an alternative cost-effective mean of simulating real fluid flows and substantially reduces lead times and costs of designs and production compared with an experimental based approach, Tiu and Liu [1]. Probably the most noticeable efforts in the field of numerical analysis of SC were made by Kovacevic et. al. [2] and [3], where, in addition to establishing a mesh procedure specific to such flow machines, the authors also explain adequate boundary calculations. Similar efforts were made by Sauls and Branch [4], 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], Steinmann [5] reported results from the modeling of a helical-lobed pump and a SC using ANSYS-CFX.

The available literature also includes several references on improvements of the compressor performance based on the analysis of the discharge port and discharge chamber, Mujic et. al. [6], Huagen et. al. [7] and Pascu et. al. [8]. In a similar manner, the investigation of the suction arrangement and inlet port was addressed by means of CFD and an optimum suction arrangement for the compressor performance was proposed based on the numerical results by Pascu et. al. in [9].

The numerical simulation of the leakage flows is the challenging aspect of running CFD in screw compressors, as the flow domain reduces in tip areas to the order of several microns. Commercial grid generators struggle in reproducing the flow path with good quality elements and user defined meshing techniques are currently available to allow the remeshing of the rotors domain to capture the modified aspect of the compression chamber at the start of each time step. This technique allows the full compressor performance prediction by means of CFD but does not allow for local mesh refinement in the tip area, where the turbulent character of the flow requires adequate boundary layer considerations. This is the area where the present paper is contributing to and several idealised flow scenarios are presented in order to suggest an optimum numerical setup for twin screw compressors.

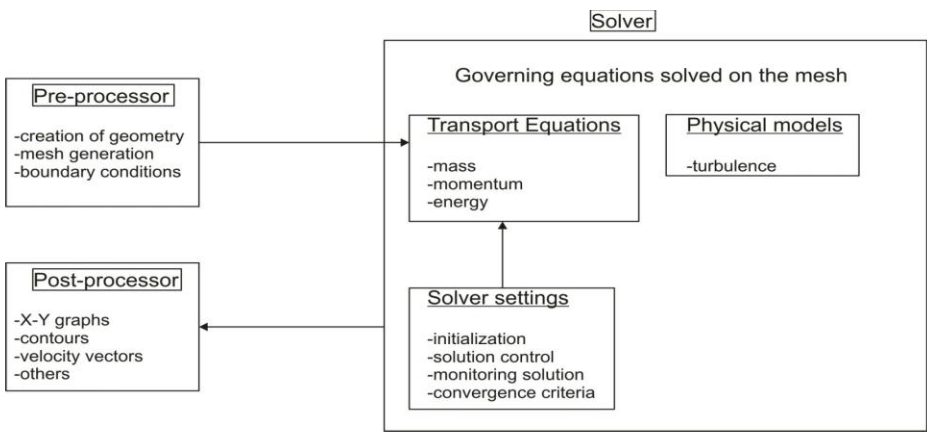

Figure 1: Standard engineering CFD simulations work flow

2. FLOW PREDICTION IN STANDARD SCREW COMPRESSORS

Running CFD simulations in most engineering fields, especially at the early design stages, has been the standard for many years now and always with good agreement to the actual tested performance.
This, however, is not the case for rotary screw compressors, where the available CFD know-how is far from being standardized, and in some respects, still in its early research stages. A typical “for engineering purposes” simulation can be described as outlined in Figure 1. In the case of a screw compressor numerical model each of these standard working blocks needs some form of user defined input.
The pre-processing stage requires the most user defined features, as standard commercial grid generators struggle to solve accurately the very tight clearances created in the working chamber.

In the following analysis, references to a standard Howden screw compressor will be made (Figure 2), where the three visible components are:

- the main casing, which hosts the two rotors in the working chamber, as well as the gas flange (suction).
- the inlet casing. An axial suction port is created at the interface between the main and the inlet casing.
- the discharge casing.

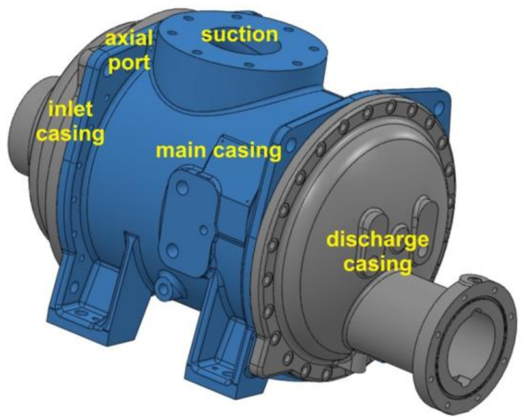

Figure 2: Compressor assembly for a standard Howden compressor

Numerical mesh

The critical sub-domains in this setup are the two rotors as they contain the working chamber as well as the clearances and leakage paths (radial, axial, interlobe and blow-hole area). Generating the grids for these domains is by far the most challenging part of the entire meshing procedure, as both micro- and macro- scale elements have to be solved. In this case, a technique dedicated to screw compressor rotors was employed, as described by Kovacević [3], which is included in SCORG© (Screw COmpressor Rotor Geometry grid generator).
A simplified representation of the rotors mesh (cross-sectional view) is presented in Figure 3.

The 3D mesh is generated by dividing the domain into 2D cross-sections along the rotor axis, calculated separately as a 2D face for both rotors, after which the following operations are implemented at each timestep:

- definition of the edges by applying an adaptive technique
- calculation of the curves connecting the facing boundaries by transfinite interpolation
- stretching function to obtain the distribution of the grid points
- orthogonalization, smoothing and final check of the grid consistency

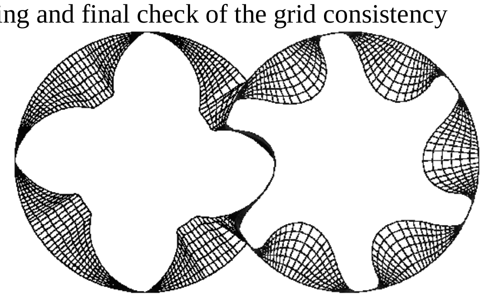

Figure 3: Mesh over rotors cross-section before 3D interpolation

This technique resulted in 143,000 nodes for the male and approximately 140,000 for the female, of structured mesh. The numerical model includes three more domains: the inlet casing, the main casing, which in turn includes the suction into the compressor and the discharge casing. ANSYS ICEM was used for the mesh generation process and special mesh refinement techniques were employed for sensitive flow areas, i.e. the interfaces with the rotors. The overall mesh statistics typically used for the compressor simulations are:

- Main casing approximately 82K nodes
- Discharge casing approximately 86K
- Inlet approximately 176K nodes

Boundary conditions

The numerical model includes the stationary domain with the major casing components (inlet, main and discharge) and the rotating domain, depicted by the two rotors, see Figure 4.

An adaptive meshing technique is utilized to capture all the changes which occur within the working chamber during the compression process. The number of time changes required by the rotors mesh is 120 for the full rotation of the male rotor, with the number of nodes kept constant across the timesteps.

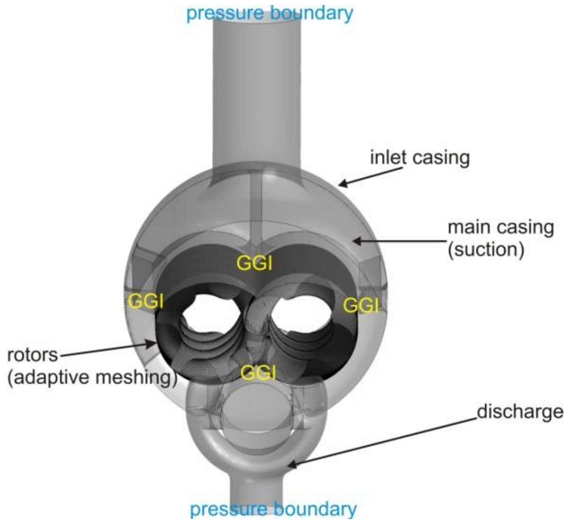

Figure 4: Standard Howden compressor numerical model

Various interfaces were applied to each of these domains to ensure the flow transition between the different domains Both the suction and the discharge were simulated by pressure boundary conditions. The pressure boundary conditions are similar to the inlet or outlet boundaries, firstly because they couple pressure and velocity directly and secondly because for all equations, apart from the momentum equation, the boundary properties are calculated from the velocity. This procedure may cause instability in the compressor cycle especially when the flow changes its direction at the boundary. This is compensated by adding a boundary domain, in which an amount of mass is added or subtracted to maintain a constant pressure, is natural and gives a stable and relatively fast solution, [3]. The compressor for air ideal gas and ran for twelve full rotations (1440 timesteps), with the first six allowing a natural pressure build-up in the system and used as initial conditions for the last six, where wall roughness was added to the rotors on top of the previous settings. The following operating conditions were investigated:

Table 1: Boundary conditions

P1	P2	n	T1	T2
[bar]	[bar]	[rpm]	[C]	[C]
1	5	3500	18	80
1	10	3500	18	80
1	15	3500	18	80

In all cases, oil injection was not simulated and standard k-epsilon turbulence was employed.

A snapshot of the fully developed flow is shown in Figure 5, depicting the streamlines progressing from the suction to the compression chamber through the axial port and exiting at the required discharge pressure through the discharge port.

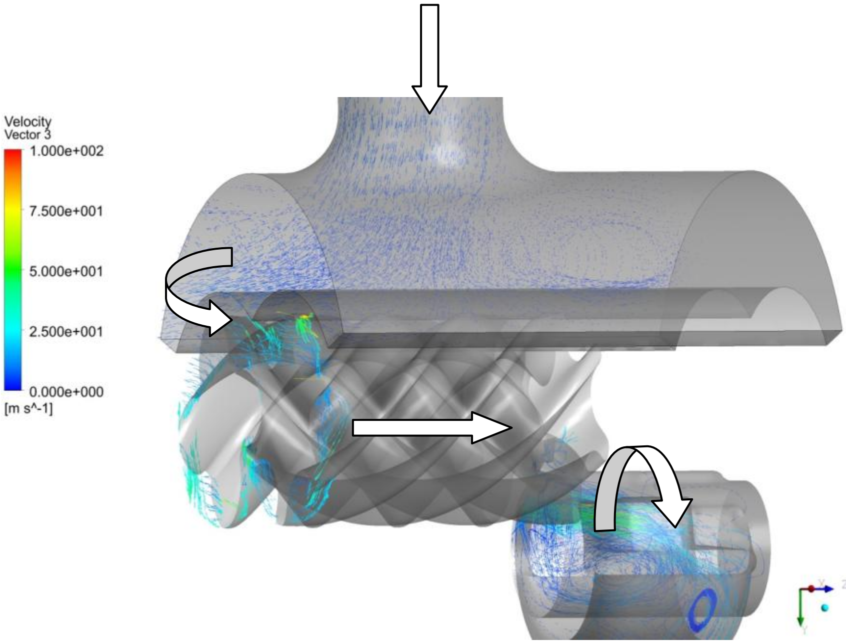

Figure 5: Fully developed flow in the standard compressor numerical model at PR=5

The compression process and internal pressure build-up are depicted for PR=5 in Figure 6. The internal pressure build-up is given by the pressure variation, at the suction and discharge ports, captured with every 360 deg (corresponding to one full rotation of the male rotor).

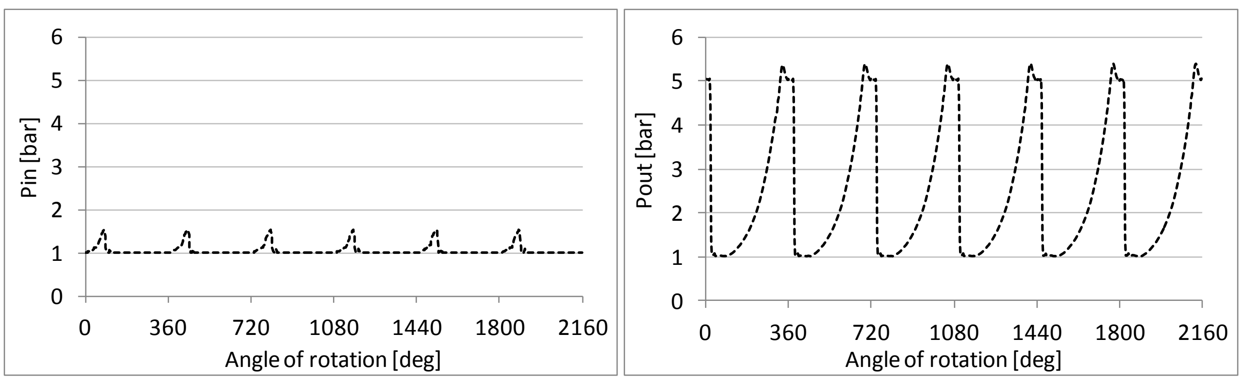

Figure 6: Pressure at the male suction and discharge end

Based on these brief observations, it can be concluded that the numerical model of the standard compressor describes well the expected flow behaviour and compression process.

The true validation of the numerical setup comes when comparing results of the CFD simulations against test measurements. Below two charts showing the power and mass flow are assessed and very good agreement between the two sets of results can be observed at PR=5 and PR=10.
However, at PR=15, the measured power is over-predicted by 8% and the mass flow is under-predicted by as much as 20%, see Figure 7.

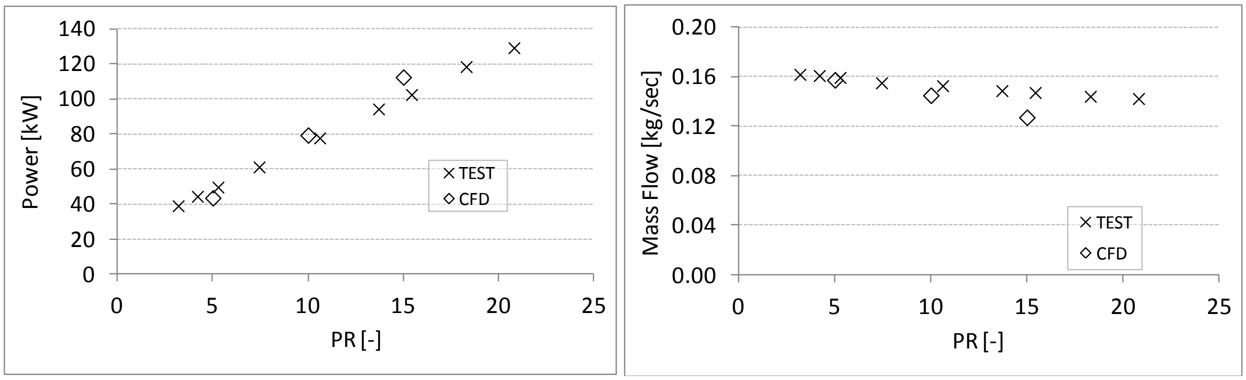

Figure 7: Standard compressor power and mass flow

With a fair amount of invalidation at the higher pressure ratios, a detailed analysis of the flow behaviour in the presented numerical setup is required, in order to understand the differences in the flow physics between the measured test data and the CFD simulations.

3. SIMPLIFIED LEAKAGE MODEL

Previously, the numerical model of a Howden screw compressor was built using a mixture of readily available pre-processing techniques, as well as a user-defined method applied for the calculation of the compression chamber.
While the validation results indicate very good agreement at smaller pressure ratios (PR=5 and PR=10), some difference in terms of both power and flow were calculated at PR=15. In an attempt to understand the source of disagreement at high pressure ratios, a “simplified leakage model” was built around the main rotor, as shown in Figure 8a. This was then sectioned along the axis of rotation in order to obtain a two-dimensional face of the leakage path around the main rotor, as indicated in Figure 8b. The highlighted two-dimensional face was then extruded along the XY plane to add a third dimension, as shown in Figure 9.

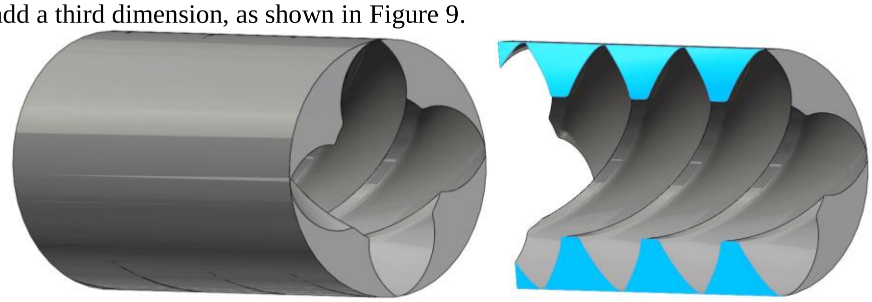

Figure 8: Flow domain around the main rotor: a) 3D model; b) 2D face along the Z-axis

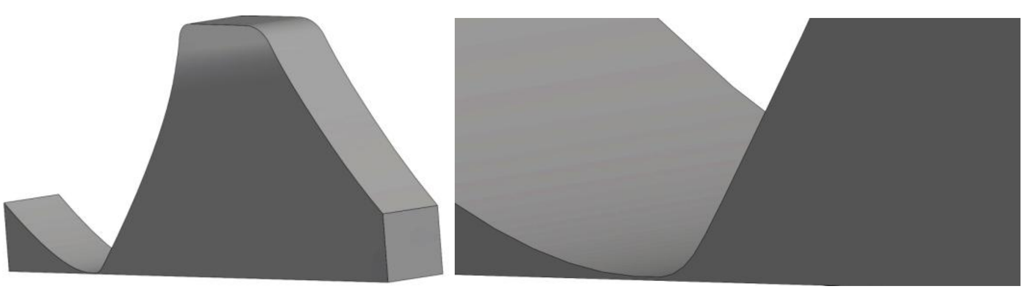

Figure 9: Simplified leakage model: a) full lobe; b) detail of the tip gap area

3.1 Mesh density

The user-defined method used to build the meshes for the two rotors was characterised by the following resolution: 30 (elements along each lobe) x 6 (divisions in the radial direction) x 30 (divisions along the Z-axis). A detail of the mesh density calculated in this manner is presented in Figure 10.

Figure 10: Mesh density in the tip gap area with 6 divisions in the radial direction

Obviously, the quality of the mesh can be improved, but results on 40 x 8 x 40 setup indicated a dramatic increase in the computational effort which resulted in non-feasible lead times for a CFD analysis of the full compressor performance curve. In order to exemplify adequate mesh requirements around the tip clearance area, several case studies with different mesh sizes and boundary layer consideration were investigated. Both structured and unstructured elements were used, as it was found that appropriate boundary layer calculations are more feasible using a tetrahedral core mesh, as shown in Figure 11. The applied boundary conditions on the simplified leakage model were derived to match the full 3D compressor model.

It was determined that the influence of appropriate boundary layer considerations has a considerable impact on the simulated flow physics, as shown in Figure 12, where differences of up to 45% were calculated between the geometries with no boundary layer ($Y+>>100$) and those with $Y+=1$ and below. This can be readily observed in the contour plots in Figure 13, where the top detail shows values of 1 all around the minimum area (maximum velocity) and the bottom plots reveal dominant peaks above 100 in the same area. $Y+<1$ values were obtained for at least 60 divisions in the radial direction, compared to 6 used in the full compressor numerical model.

A low-Reynolds k-epsilon model would typically require a near wall resolution of $y+<0.2$, while a low-Reynolds number k-omega model would require at least $y+<2$. In industrial flows, even $y+<2$ cannot be guaranteed in most applications and for this reason, a new near wall treatment was developed for the k-omega models. It allows for smooth shift from a low-Reynolds number form to a wall function formulation. This is aspect will be fully addressed in the next section.

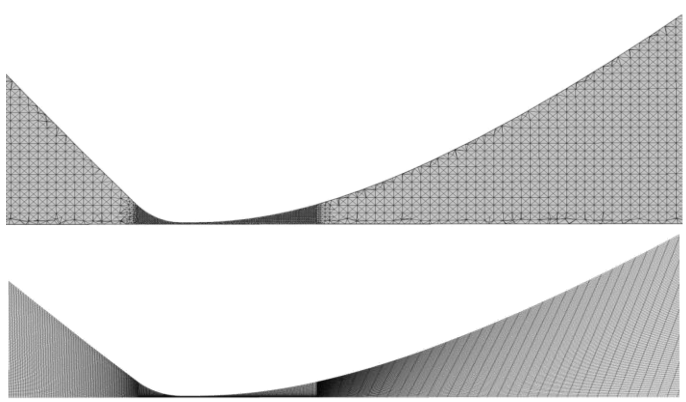

Figure 11: Mesh density requirements in the tip clearance area: top – with tetrahedral elements; bottom – with hexagonal elements

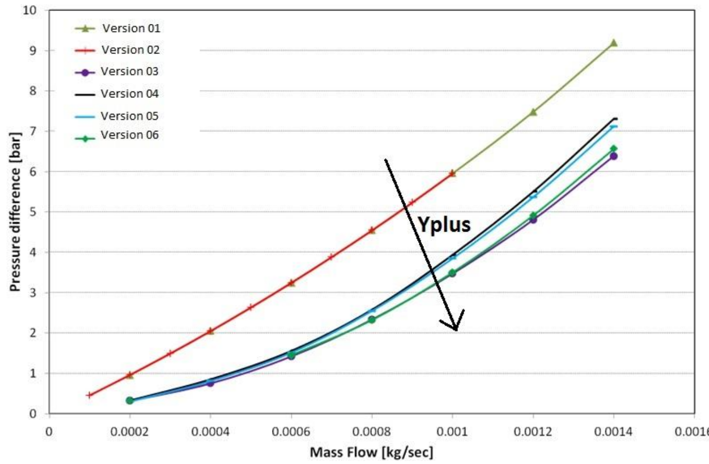

Figure 12: Mesh density requirements in the tip clearance area

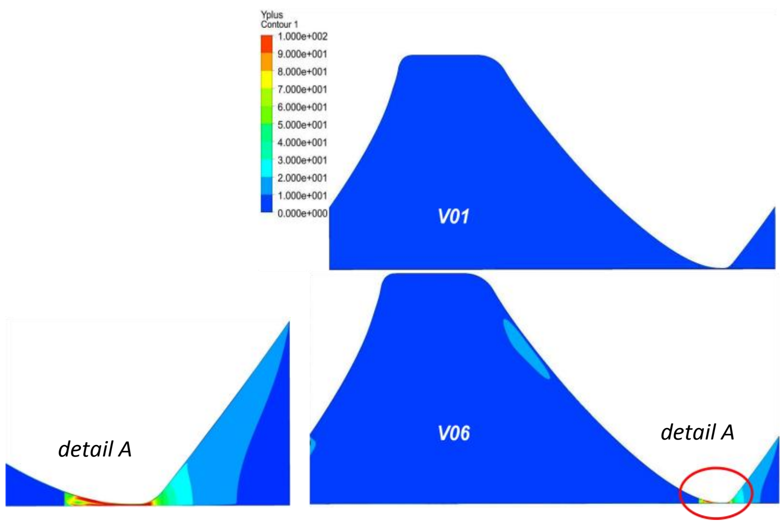

Figure 13: Y+ contour plots

3.2 Turbulence modelling

One of the main problems in turbulence modelling is the accurate prediction of flow separation from a smooth surface. Standard two-equation turbulence models often fail to predict the onset and the amount of flow separation under adverse pressure gradient conditions. In general, turbulence models based on the epsilon -equation predict the onset of separation too late and under-predict the amount of separation later on, [11]. The prediction is therefore not on the conservative side from an engineering stand-point.

In order to quantify the amount of flow separation occurring in the simplified leakage model, two different scenarios were simulated for the mesh with Y+<1, i.e. k-epsilon and SST. Results are presented in Figure 14.

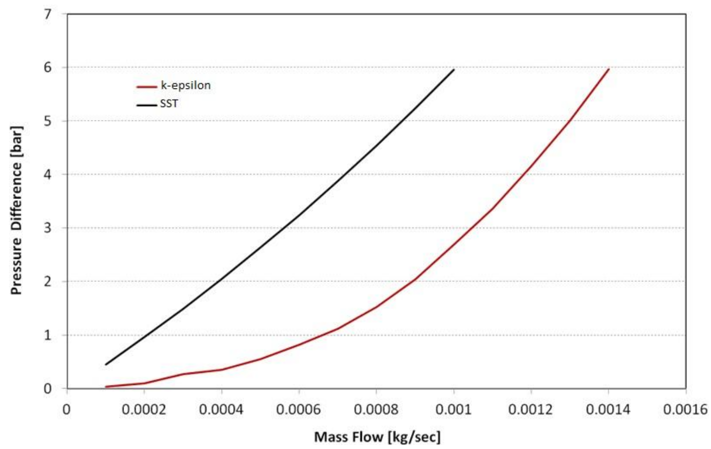

Figure 14: Turbulence modelling of the simplified leakage model

It can be readily observed that at any given value of the mass flow rate, the predicted pressure difference by the k-epsilon model is severely underestimated when assessed against the equivalent SST result. Increasing mass flow values in the above diagram are representative for the higher pressure ratios simulated in the previous sections, where the invalidation of the numerical results of the standard screw compressor model was at its highest against the test data. This can now be explained by the choice of turbulence model.

One of the well known deficiencies of the k-epsilon model is its inability to handle low turbulent Reynolds number computations. Complex damping functions can be added to the k-epsilon model, as well as the requirement of highly refined near-wall grid resolution in an attempt to model low turbulent Reynolds number flows, but this approach often leads to numerical instability. Some of these difficulties may be avoided by using the k-omega model, making it more appropriate than the k-epsilon model for flows requiring high near-wall resolution. However, a strict low-Reynolds number implementation of the model would also require a near wall grid resolution of at least $y+ < 2$. This condition cannot be guaranteed in most applications at all walls and for this reason, the automatic new near wall treatment available in the SST offers an improved prediction of the onset and the amount of flow separation under adverse pressure gradients by the inclusion of transport effects into the formulation of the eddy-viscosity, as presented in [12]. As a consequence, SST turbulence model is found more accurate for the nature of the flow present in the leakage area of a screw compressor, but with some expense at the required computational resources.

4. CONCLUSIONS

The performance of twin screw compressors is highly affected by leakages, which are dependent on various clearances and the pressure differences across these clearances. The numerical simulation of the leakage flows is the challenging aspect of running CFD in screw compressors, as commercial grid generators struggle in reproducing the flow path with good quality elements. One user-defined meshing technique is currently available to allow the remeshing of the rotors domain to capture the modified aspect of the compression chamber. This technique allows the full compressor performance prediction by means of CFD and probably reflects the state-of-the-art in terms of the numerical investigation of twin screw compressors.

The numerical model of a Howden screw compressor was built using a mixture of readily available pre-processing techniques, as well the user-defined method applied for the calculation of the compression chamber.
When validating against actual test measurements, it was observed that the proposed setup delivered a very good agreement in terms of power and mass flow at smaller pressure ratios. However, at higher pressures, the measured power is over-predicted by 8% and the mass flow is under-predicted by as much as 20%,

With a fair amount of invalidation at the higher pressure ratios, a detailed analysis of the flow behaviour in the presented numerical setup was carried out, in order to understand the differences in the flow physics between the measured test data and the CFD simulations.

At this point in the analysis, it was observed that the user-defined meshing technique employed for the two rotors does not allow for local mesh refinement in the tip area, where the turbulent character of the flow requires adequate boundary layer considerations. For this, a simplified leakage model was proposed, where in order to exemplify adequate mesh requirements around the tip clearance area, several case studies with different mesh sizes and boundary layer consideration were investigated. It was determined that the influence of appropriate boundary layer considerations has a considerable impact on the simulated flow physics, as shown in Figure 14, where differences of up to 45% were calculated between the geometries with no boundary layer (Y+>>100) and those with Y+<1. It was concluded that an improvement of at least one order of magnitude in the number of mesh elements in the tip clearance area is required in the user-defined technique used to mesh the three-dimensional model of the screw rotors. This however, is not possible due to the increase in the computational effort which results in non-feasible lead times for a CFD analysis of the full compressor performance curve.

Another element identified as contributing cause to the invalidation at higher pressure ratios was the choice of turbulence model, as the SST model was proven to deliver more accurate predictions of the flow physics in the simplified leakage model, although it is not preferable for the three-dimensional model due to its impact on the lead computational time. This model SST offered an improved prediction of the onset and the amount of flow separation, where k-epsilon was found insufficient for appropriate boundary layer considerations.

The combined influence of mesh density requirements and appropriate turbulence modelling was found paramount for the user-defined meshing technique employed in this paper and the authors suggest any further development around using CFD in screw compressor models to include such considerations.

REFERENCES

[1] Tu, J., Yeoh, G. H., and Liu, C., 2008, "Computational fluid dynamics", Butterworth – Heinemann
[2] Kovacević, A., Stosic, N., and Smith, I., "CFD analysis of screw compressor performance", Centre for positive displacement compressor technology, City University, London
[3] Kovacević, A., Stosic, N., and Smith, I., "Screw compressors. Three dimensional computational fluid dynamics and solid fluid interaction", Springer Verlag, 2007
[4] Sauls, J., and Branch, S., 2009, "CFD analysis of refrigeration screw compressors", Ingersoll Rand
[5] Steinmann, A., 2006, "Numerical simulation of fluid flow in screw machines with moving mesh techniques in ANSYS CFX", Schraubenmaschinen 2006; VDI Verlag GmbH
[6] Mujic, E., Kovacević, 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] Huagen, W., Xing, Z., Peng, X., and Shu, P., "Simulation of discharge pressure pulsation within twin screw compressors", Journal of Power and Energy, IMechE, 2004
[8] Pascu M., Kovacevic A., Udo N., 2012, "Performance Optimization of Screw Compressors Based on Numerical Investigation of the Flow Behaviour in the Discharge Chamber", Proc Int Compressor Conf at Purdue, Purdue, pp. 1145
[9] Pascu, M., Heiyanthuduwage, M., Mounoury, S., Cook, G., and Kovacevic, A., "A study on the influence of the suction arrangement on the performance of twin screw compressors", ASME IMECE 2013 Paper IMECE 2013-62391
[10] Stosic, N., Smith, I., and Kovacević, A., "Screw compressors. Mathematical modelling and performance calculation", Springer Verlag, 2005
[11] Jovanović, J., "The statistical dynamics of turbulence", Springer Verlag, 2004
[12] Menter, F.R., "Zonal two-equation turbulence models for aerodynamic flows", AIAA Paper 96-2906, 1993
[13] Guerrato, D., Nouri, J. M., Stosic, N., Arcoumanis, C., and Smith, I., "Flow development in the discharge chamber of a screw compressor", Centre for positive displacement compressor technology, City University, London
[14] http://www.ansys.com/Products/Simulation+Technology/Fluid+Dynamics/ANSYS+CFX