2000 International Compressor Engineering Conference at Purdue,
Purdue University, West Lafayette, Indiana, 25-28 July 2000

THE CFD ANALYSIS OF A SCREW COMPRESSOR SUCTION FLOW

A. Kovacevic, N. Stosic and I. K. Smith

Centre for Positive Displacement Compressor Technology,
City University, London EC1V OHB, U.K,
n.stosic@city.ac.uk, www.city.ac.uk/staff/~sj376

ABSTRACT

Dynamic flow losses in the suction chamber play a very important if not the main role in the efficiency decrease in screw compressors. The design of these machines, together with the accounting of flow losses and how to reduce them, is still based only on the simple analysis. Since Computational Fluid Mechanics offers today a more accurate estimation of the velocity, pressure, temperature and concentration fields within the screw compressor, such studies can be thus extended to design a suction port with the minimized flow losses. This paper presents application of a CFD procedure to calculate a screw compressor suction flow. The numerical grid of a suction chamber is generated by a 2-D transfinite interpolation combined with the layer meshing technique both facilitated by use of an independent and stand-alone interface program which connects a screw compressor geometry with a conventional numerical pre-processor. The calculation is applied to the oil-flooded screw compressor and obtained with Comet, a commercial CFD solver.

INTRODUCTION

The performance of screw compressors may be predicted reasonably well and further performance improvements can be made either in oil injected or oil free types of machines by use of a flow simulation models based on Computational Fluid Dynamics (CFD). There is a large number of publications in the CFD field, such as those of Demirdzic and Peric, 1990, Demirdzic et al, 1993 and Ferziger and Peric, 1996. However, only several papers have been published on the use of this technique for the analysis of flow through screw compressors, for example, by Stosic et al, 1996 and Kovacevic et al, 1999. This is mainly due to the complexity of both geometric configuration and the flow through screw compressors. The magnitude of screw compressor dimensions varies from several centimetres for the compression chamber to few micrometers for the clearances. Therefore a suitable numerical grid must be generated, which not only accurately describes the screw compressor geometry, but also can handle its associated stretching and sliding motion. Standard pre-processors are still insufficiently developed to do this. The first requirement for the use of a numerical solver is therefore to develop an interface, which will enable screw compressor geometry to be used within standard codes. A solver applied should be able to cope with high acceleration, phase change, particles and turbulence, all of which are important elements in screw compressor flow. Some CFD codes are already available which seem to be able to cope with screw compressor geometry, but they still need substantial development to obtain useful results with them. Once obtained, results of the CFD analysis are found extremely useful because the flow patterns, which indicate the process in the suction chamber than can then be used as a powerful facility to test and validate any suction geometry mode for different working conditions and to optimize a suction port.

A NUMERICAL GRID OF A SCREW COMPRESSOR SUCTION CHAMBER

In order to use a CFD solver based on a finite volume method to solve flow equations, a spatial domain is replaced by a mesh which contains discrete volumes. The number of these volumes depends on the problem dimensionality and accuracy required. A structured and composite grid, made of several partially discontinuous grids patched together and based on a single boundary fitted co-ordinate system is used to transform the compressor geometry into discrete elements. Grids are then connected over regions on their boundaries coinciding with other parts of the entire numerical mesh.

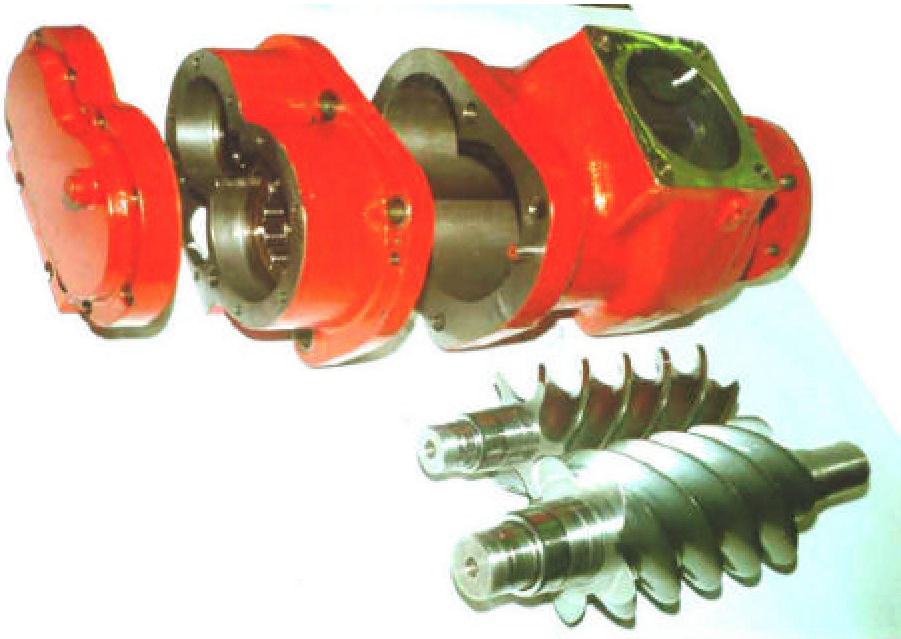

Fig 1 Screw compressor with 'N' Rotors used in the numerical example

A screw compressor which geometry is used in the numerical example is presented in Fig. 1. Its suction port is on the top of the compressor and it is defined between the rotors and compressor housing. The geometry of the suction port is subdivided into 5 simple grid domains and shown in Fig. 2 together with the characteristic section planes used to present 3-D results within the 2-D domains. The 5/6 'N' rotors, of the outer diameter of the main rotor 127.45 mm, L/D=1.65, which are a part of the computational domain, have their own spatial regions defined by the rotor profile coordinates. These were calculated by means of the rack generation procedure described by Stosic et al, 1997. The grid portions thus defined, determine all connections between the rotors and the housing and include the interlobe, tip and blow-hole leakage paths. The mesh calculation was based on an algebraic transfinite interpolation procedure. This included stretching functions, to ensure grid orthogonality and smoothness. An iterative procedure was used to obtain a satisfactory grid distribution.

The procedure described is used in a stand alone CAD - CFD interface to produce a regular 3-D grid of the screw compressor suction chamber. The interface program, written in Fortran, calculates the meshing rotor coordinates from the given rotor or rack point coordinates, then calculates grids for both rotors. It also calculates grids for the compressor housing and prepares the control parameters necessary for the CFD calculation of the compressor fluid flow and its post processing.

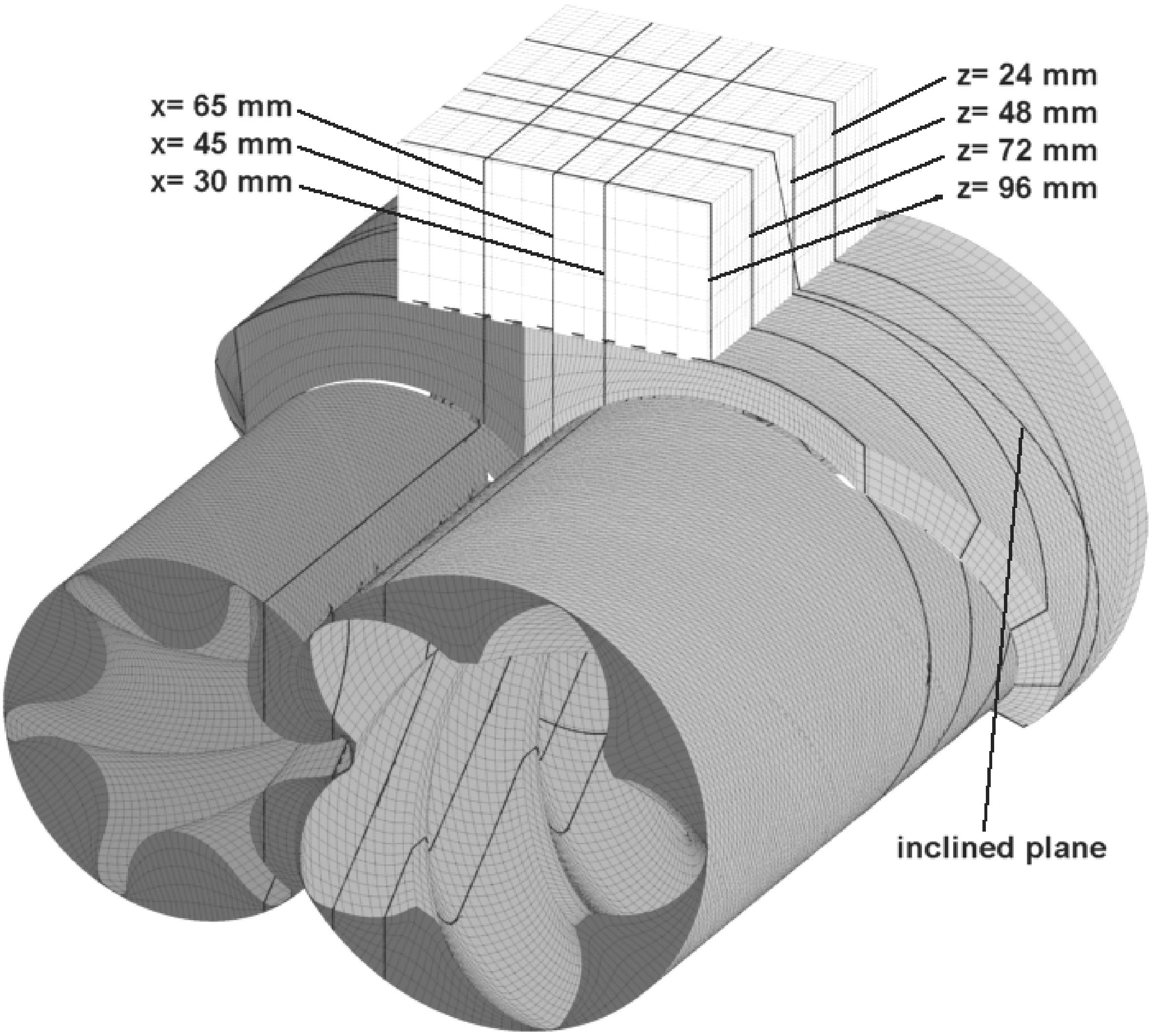

Fig 2 Computational domain of the screw compressor suction chamber with section planes

A transfer file is written in ASCII format and includes the node and cell definitions, regions, boundary conditions, control parameters and post-processing function parameters. This enables a full flexibility allowing the file to be imported into any commercial CFD package through their standard pre processors which themselves must be capable to process control volumes with an arbitrary number of faces. The solution domain is then split into several regions, with separate grid generation in each of them, without the need for grid matching at the region interfaces. Despite the non-matching interfaces, the discretization method is fully conservative and all regions are coupled, so that the solution converges well as if the grid were made with one pass only. This is very convenient when grids of different topology are generated to obtain the best fit for the geometry of each region. It is then possible to achieve high grid density without the large deformation that would result from a single-region grid. The grid can also be refined locally by subdividing selected cells into a number of smaller cells. The fact that the control volumes may have any number of faces and that the grids do not have to match at interfaces makes it possible to compute flows where the grid moves in some regions while it remains stationary in others. More details of the grid types used and the relative advantages of each grid system and on the interface are given in full detail by Kovacevic et al, 2000. The grid used for the present calculation consists of 5 independent regions on 250,000 volume cells.

RESULTS OF A FLOW CALCULATION IN THE COMPRESSOR SUCTION

Fluid, which is compressed within a screw compressor, can be in the form of a gas, a vapour or a wet mixture of liquid and vapour. The densities of all of these vary with pressure and temperature. The compressor flow is fully described by the mass averaged equations of continuity, momentum and energy equations, which are accompanied by the equations of k-ε turbulence model and equation of state, as given in detail by Stosic et al, 1996.

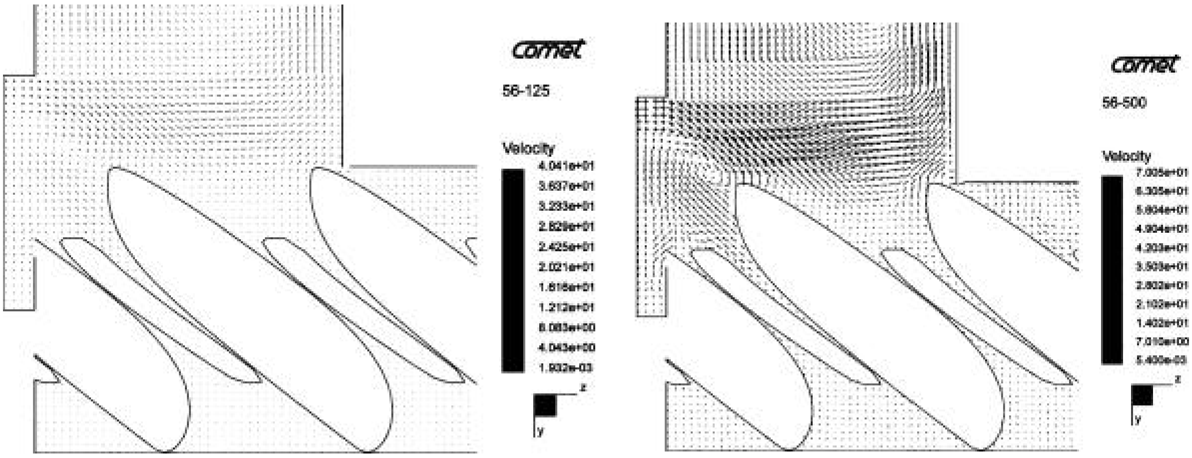

Fig 3 Velocity vectors in the axial section x=45 mm

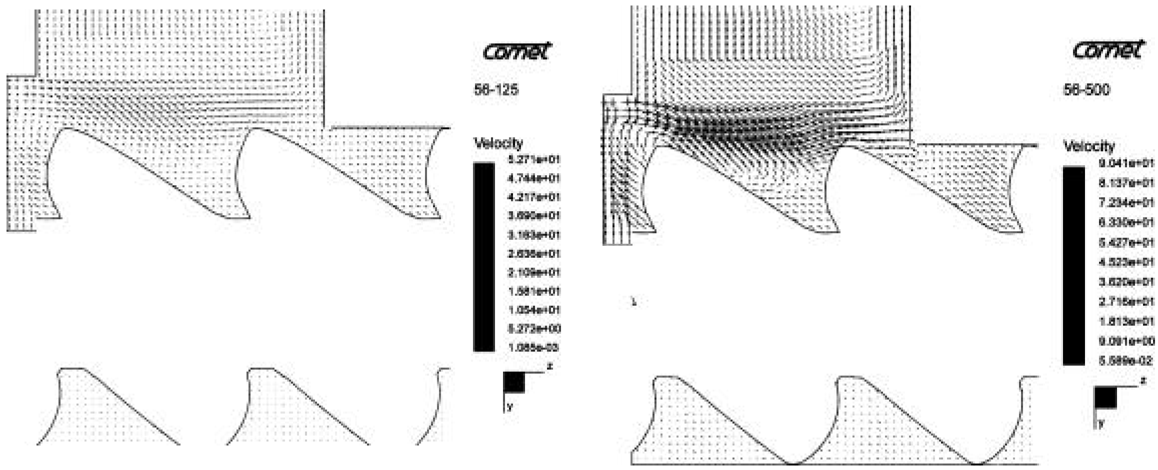

Fig 4 Velocity vectors in the axial section x=30 mm

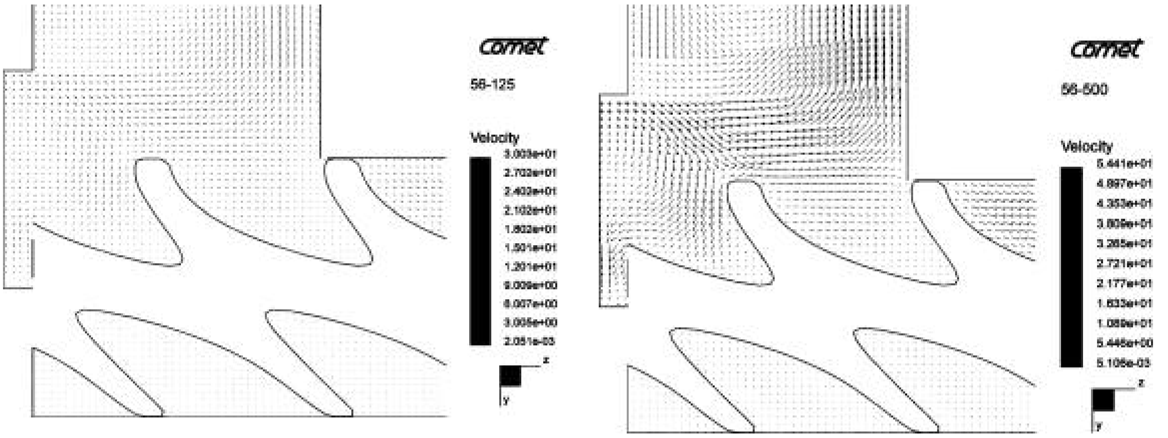

Fig 5 Velocity vectors in the axial section x=65 mm

A numerical solution of the equation system was obtained by the CFD solver ‘Comet’ which is developed by the ICCM Institute of Computational Continuum Mechanics GmbH, Hamburg, Germany. This code is applicable not only to calculate a fluid flow, but also either for solid body analysis or for a fluid – solid interaction. It also meets the needs of coupled computation of gas and liquid flow including moving surfaces, non-Newtonian fluids, flows with both highly compressible as well as incompressible regions, flows with moving boundaries and particle flows. Hence, it contains all the essential features needed to obtain the ultimate aims of full screw compressor simulation.

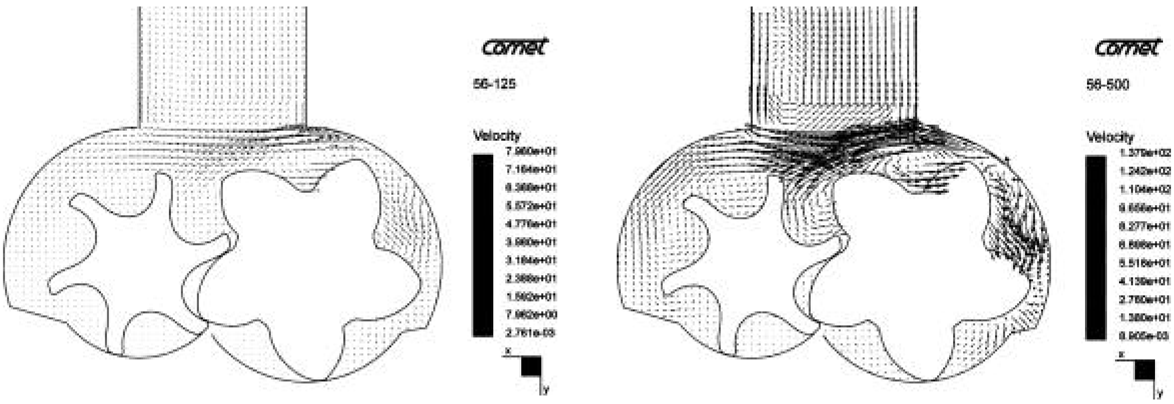

Fig 6 Velocity vectors in the cross section z=24 mm

The adopted procedure was applied to an oil-flooded screw compressor of medium size and average working parameters, the same which was described by Stosic et al, 1997. Some results of flow calculations thus obtained are presented to demonstrate the capabilities of the procedure adopted. The flow field is presented as a comparison of velocity vectors in one axial and five cross sections for two different compressor rotational speeds, 125 and 500 rad/s.

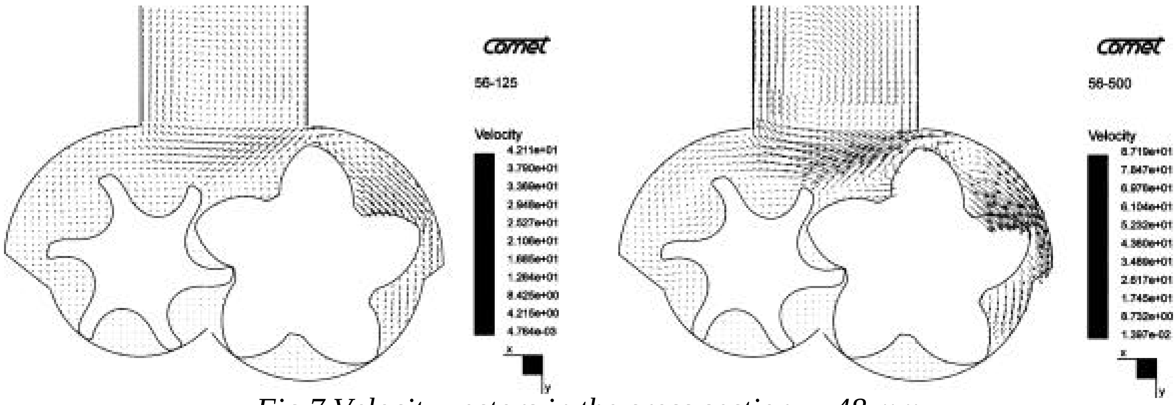

Fig 7 Velocity vectors in the cross section z=48 mm

Velocity vectors in the axial plane are given in Fig. 3. For a position of the axial section, refer Fig 2. This interesting section reveals cuts of the rotors in the cusp-to cusp axial section and the flow between them. Not only is a flow pattern in the working chamber presented in Fig 3, but also the flow through the interlobe clearances.

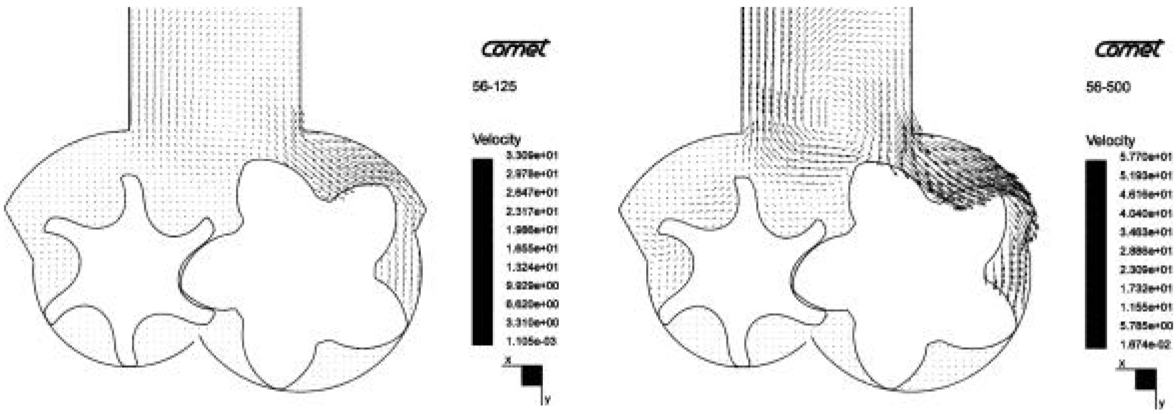

Fig 8 Velocity vectors in the cross section z=72 mm

Velocity vectors in the cross sections, which consecutively cut the rotors and suction chamber, are plotted in Figs 4-7. These cross sections are also defined in Fig. 2.

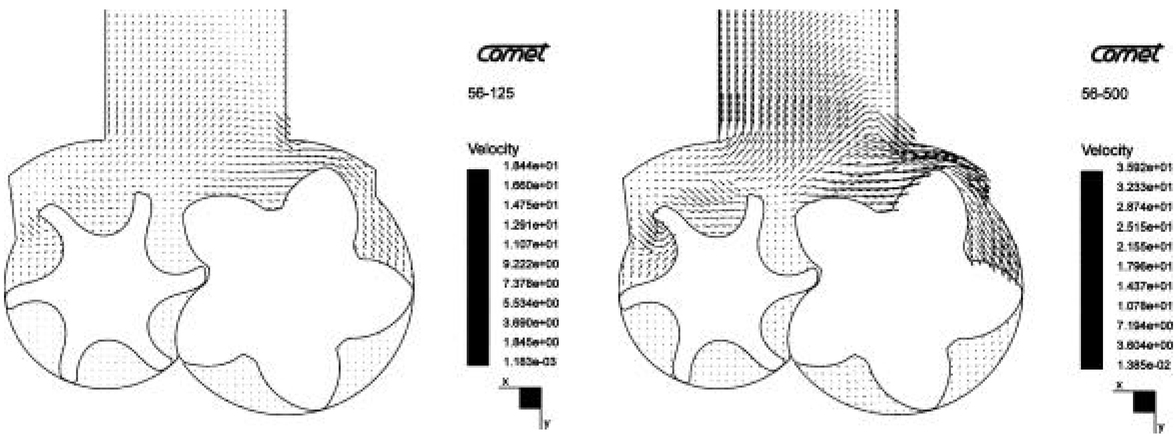

Fig 9 Velocity vectors in the cross section z=96 mm

As indicated in Fig 2, a section inclined in reference to the rotor axes, which is presented in Fig 8, shows velocity vectors in a special position, which indicates that the flow is fully reversed if compared to the flow in the cross sections.

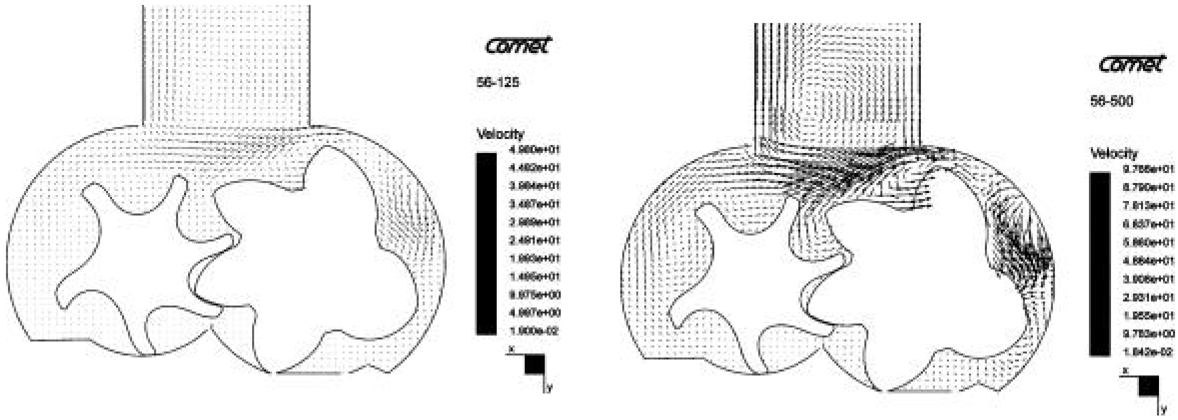

Fig 8 Velocity vectors in the section inclined to the rotor axes

The fluid flow in the compressor suction is mainly responsible for a majority of dynamic flow losses in a screw compressor. These are due to high fluid velocity in certain regions and recirculation between the rotors and the compressor housing. The flow patterns presented are only one of a number of different situations which may arise in a compressor. By changing the input parameters, like rotational speed, suction flange size and suction shape, a thorough analysis of their influence upon the overall compressor process may be performed.

CONCLUSIONS

Computational Fluid Dynamics is frequently used for calculation of fluid flows in stationary and rotating machines and special codes have been developed specifically for this purpose. In the authors' opinion the small number of screw compressor flows published in the literature is mainly due to the complexity of the screw compressor configuration but also because of the complexity of the physical processes which include fluid density variation as well as two-phase and particulate flows.

To bridge the gap between the screw compressor geometry and CFD code a stand alone interface has been developed in Fortran which enables a user to transfer the rotor and other screw compressor geometry data directly into various CFD codes. The only prerequisite is that the screw compressor geometry is given in a general ASCII or binary format and that the CFD ASCII or binary input format is known.

An example of the oil-free screw compressor has been given to demonstrate the scope of the method for accurate calculation of processes within screw compressors.

REFERENCES

Demirdzic I, Peric M, 1990, Finite Volume Method for Prediction of Fluid flow in Arbitrary Shaped Domains with Moving Boundaries, *Int. J. Numerical Methods in Fluids* Vol 10, 771

Demirdzic I, Lilek Z, Peric, M, 1993, A Collocated Finite Volume Method for Predicting Flows at All Speeds, *Int. J Numerical Methods in Fluids*, Vol. 16, 1029

Ferziger J H, Peric, M, 1996, Computational Methods for Fluid Dynamics, *Springer, Berlin*

Kovacevic A, Stosic N, Smith I. K, 1999, Development of CAD-CFD Interface for Screw Compressor Design, *International Conference on Compressors and Their Systems, London, ImechE Proceedings*, 757

Kovacevic A, Stosic N, Smith I. K, 2000, Grid Aspects of Screw Compressor Flow Calculations, *ASME Congress, Orlando FL*

Stosic N, Smith I. K. and Zagorac S, 1996, CFD Studies of Flow in Screw and Scroll Compressors, *XIII Int. Conf on Compressor Engineering at Purdue*, July 1996

Stosic, N, Smith, I. K, Kovacevic, A and Aldis, C. A, 1997, The design of a twin-screw compressor based on a new rotor profile. *Jrnl of Engng Design*, v.8, n.4, 1997, pp389-399