NUMERICAL INVESTIGATION OF A SCREW COMPRESSOR PERFORMANCE

Ion MĂLĂEL[5], Mihail SIMA[5]

ABSTRACT: In this paper the CHP 220 screw compressor performance evaluation was performed by using the numerical methods. Thus, for the CFD analysis has defined the computing domain comprised of two sub-domains, one rotational and one stationary. The TwinMesh software, dedicated to volumetric machines with positive displacement, was used for the meshing of the rotors. The SST turbulence model has been used for flow modelling. Also, the finite element analysis of the stress and deformation state in the screw compressor rotors was performed. As inputs to FEM analysis, the results of the CFD analysis, performed on the working environment, were used. Finally, the results consist of the variation of the absolute pressure from the inlet to the compressor outlet, the variation of the massflow, the distribution of the sealing fluid (oil), but also the distributions of the stress tensor and the displacements together with the safety coefficients.

KEYWORDS: screw compressor, CFD, FEM, TwinMesh, volumetric machine

NOMENCLATURE

All variables are defined throughout the work.

1. INTRODUCTION

The screw compressors are volumetric machines with positive displacement that can be used in refrigeration and air conditioning but also in power generation. Even though the concept of screw compressor is relatively new, studies have been developed on thermodynamics of screw compressors in order to obtain a calculation model. These models were used both for performance evaluation and optimization of rotor lobe profiles. Among the best-known models are those made by Bein and Hamilton [1], Boblitt and Moore [2], Jianhua and Guangxi [3]

CFD (Computational Fluid Dynamics) methods can be used to predict the performance of such machine. The use of these methods in studying flow in screw compressors involves studies of non-stationary flow with movement boundaries. Flows with such boundary were also developed by Peric [4], Demirdzic and Peric [5], and Demirdzic and Muzaferija [6].

When using finite volume methods in studying screw compressor flow, a particularly important problem, namely generating the calculation grid for unsteady simulations with grid movements, appears. As the volumes in the compressor change as the rotors rotate, the calculation grid must be deformed. At present, no commercial computing grid is able to do so, as Kovacevic [7] and Prasad [8] have shown in their works.

Despite the above, there are only a few reports on unsteady flow in 3D for screw compressors with high discharge pressures.

Increasing the storage capacity of natural gas is one of the directions of action adopted by the main energy suppliers. Achieving the objectives following this action line requires the development of compression capabilities with direct implications in increasing the working pressure of the compressors. In this regard, the design of a screw compressor to provide a high flow rate at a considerable discharge pressure implies the evaluation of the mechanical strength of the compressor parts, given that the compressor standards provide a test sample of the assembly at a pressure of 1,5 times the discharge pressure. The rigidity evaluation will be

[5] National Research and Development Institute for Gas Turbines COMOTI, Bucharest, Romania

done by calculating the deformations generated by the pressure and distribution of the thermal field on the rotors.

In this study, the CHP 220 screw compressor having a suction pressure of 4.5 bar and 30 bar at discharge.

2. PAPER CONTENTS

By using the CHP 220 screw compressor geometry, we have defined the computational domain for this study. Figure 1 shows the geometry of the screw compressor used in numerical analysis.

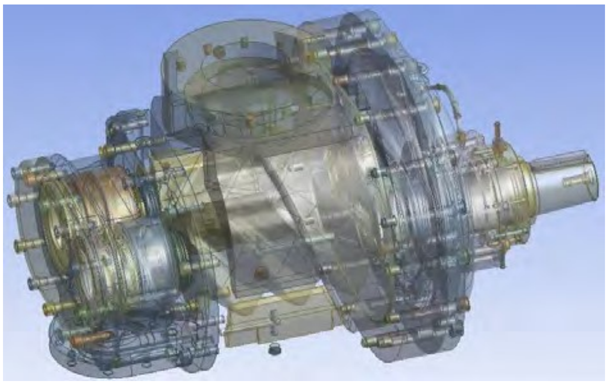
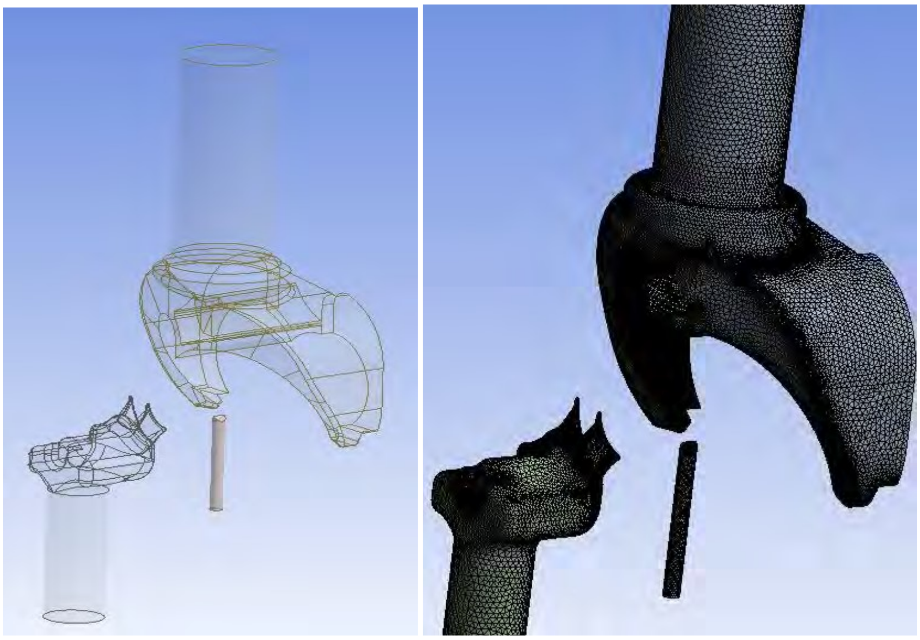


Fig. 1. The CHP 220 screw compressor geometry

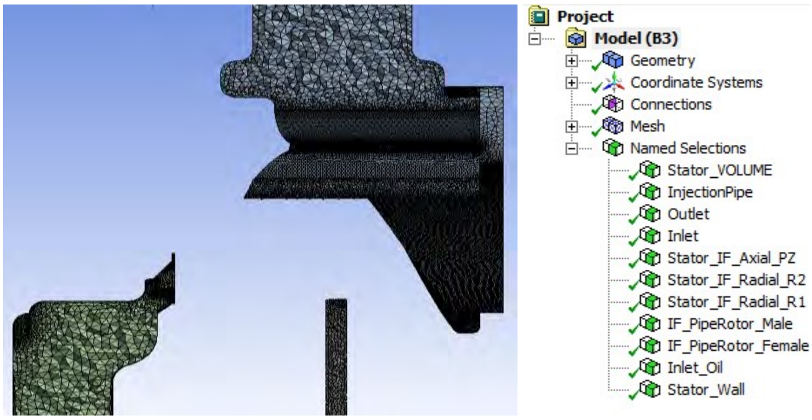
Starting from the 3D CAD model, the computational domain for the stator component of the screw compressor, which is composed of three subdomains, has been defined: the suction sub-domain, the oil input sub-domain, and the sub-domain of the discharge. Boolean functions for subtraction and union were used to define the domains. In figure 2 a) are represented the three subdomains made using the Ansys Design Modeler software.

The mesh (Figure 2b) for the stator component was generated with Ansys Meshing, where meshing methods were used based on the surface size and volume size of the element. Figure 3 shows the surfaces for the boundary conditions and a cross section of this mesh.



a) b)

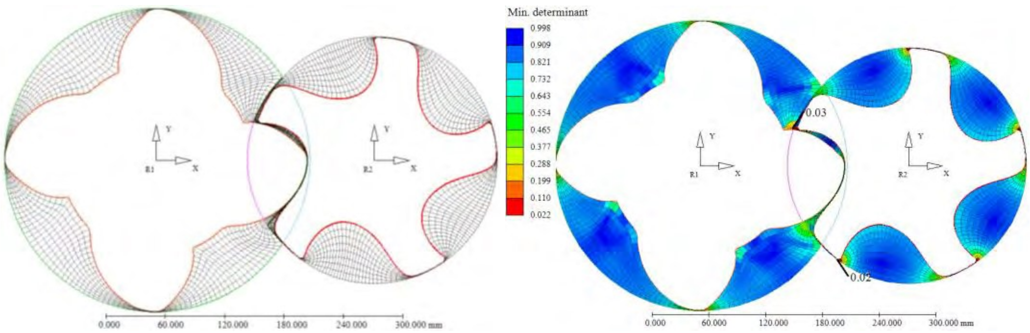
Fig 2. a) Computational domain for the stator parts; b) The mesh for stator parts



a) b)

Fig. 3. a) Cross section thru the mesh; b) Surfaces selection for BC

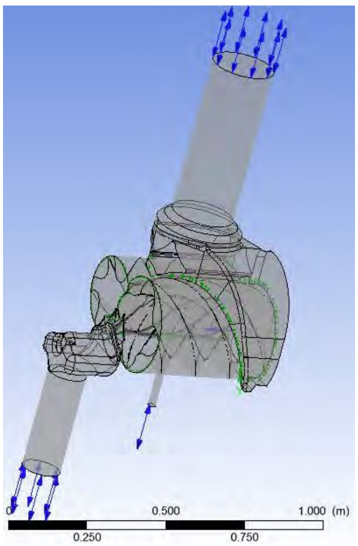
For the rotor domain, the commercial software for volumetric machines with positive displacement, TwinMesh, was used. Figure 4 shows the 2D grid and its quality.



a) b)

Fig 4. a) 2D mesh; b) Mesh quality

Figure 5 shows the boundary conditions defined in Ansys CFX Pre together with the list of expert parameters used for solution stability. For this analysis, the SST turbulence model was used. As boundary conditions were used opening conditions where the pressure value was imposed. For the outlet a function for increasing the pressure has been defined.




```
FLOW: Flow Analysis 1
  &replace EXPERT PARAMETERS:
   backup file at zero = t
   compressible linearisation option = 3
   max solver its fluids = 80
   min solver its fluids = 40
   multigrid solver = f
   outer loop relaxations default = 0.1
   solver relaxation fluids = 0.8
   solver target reduction fluids = 0.05
   stationary interface reintersection = f
   topology estimate factor = 1.2
   topology estimate factor zif = 1.2
  END
END
```

a) b)

Fig 5. a) BC; b) Expert parameters from ANSYS CFX pre

In the figure 6 is represented the absolute pressure from the compressor. The blue color is the suction pressure value, 4.5 bar, and the red color is the discharge pressure that is 30 bar. Using the "opening" boundary conditions with the required pressure, both the massflow variation (Figure 7a) and the variation of the oil flow at the compressor inlet (Figure 7b) were determined. Also the power variation for a complete 360 degree rotation was determined (Figure 8).

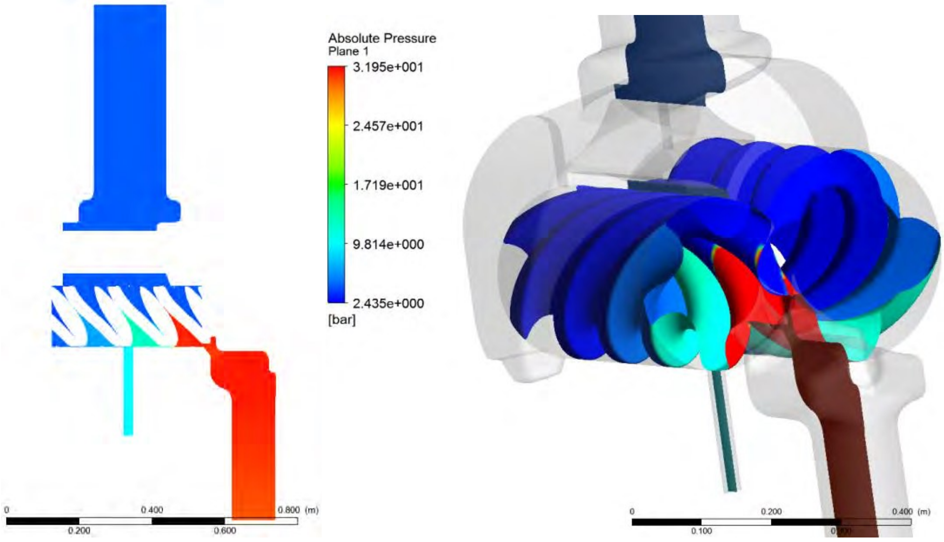
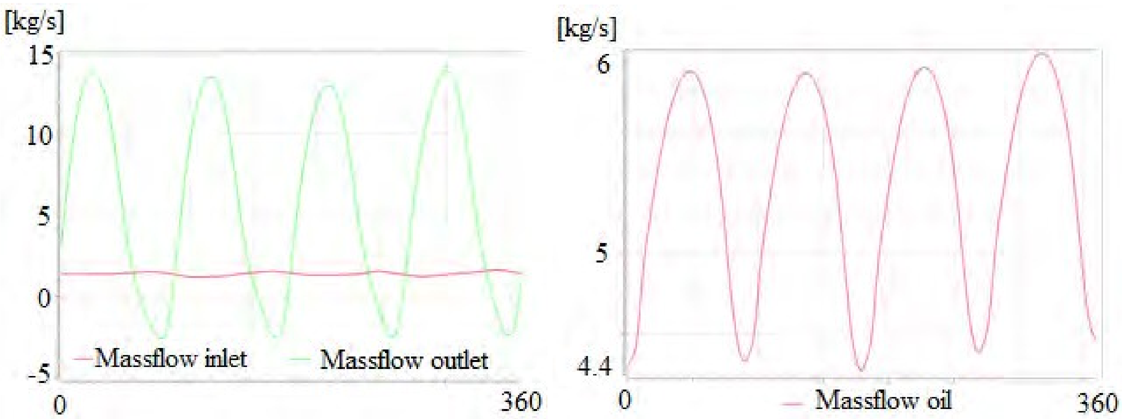


Fig 6. Absolute pressure



a) b)

Fig 7. a) Air massflow; b) Oil massflow

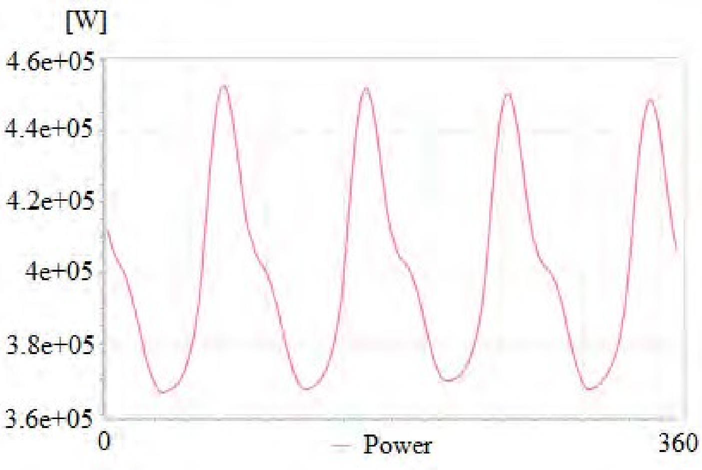
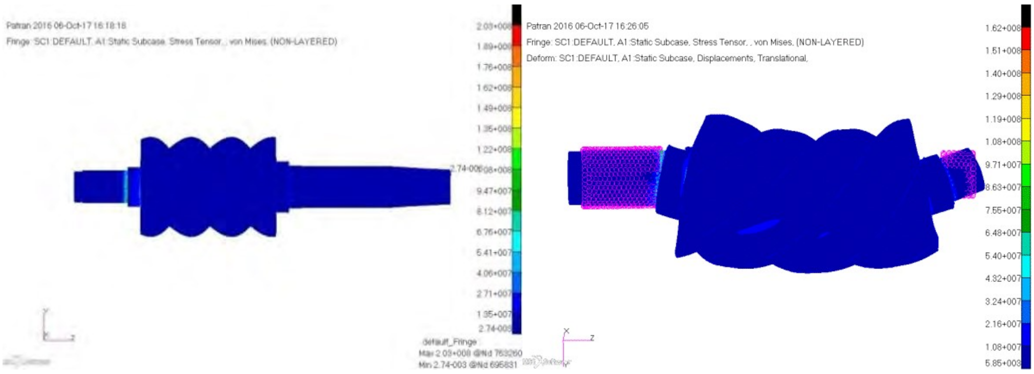


Fig 8. Power variation

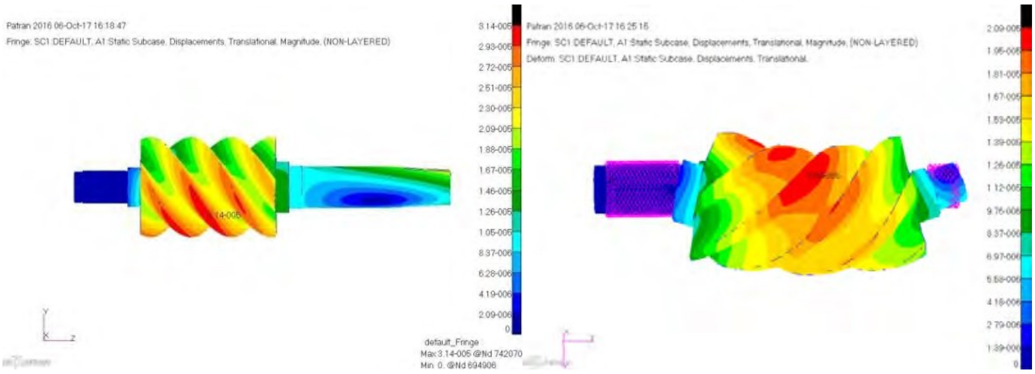
For FEM analysis the following conditions were used: - inlet pressure of 0.45 MPa; - discharge pressure of 3.0 MPa; - maximum working pressure 4,1 MPa, and the mechanical properties of the material were as follows: elastic modulus: E = 1.68E11 N / mm2; Poisson's coefficient: ν = 0.28;   = 0.28; density ρ = 7860 kg / m3; yield limit: σ_Y = 275 MPa; break elongation: ε_r = 22%; ultimate stress: σ_U = 475 MPa.

The Von Misses stress for both, male and female rotor is presented in figure 9 and the displacements in figure 10.



a) male b) female

Fig 9. The Von Misses stress



a) male b) female

Fig 10. Rotors displacements

3. CONCLUSIONS

Numerical performance of a screw compressor using numerical methods was evaluated in this paper. For this study the geometry of the CHP 220 screw compressor has been used, which has a pressure of 4.5 bar at the inlet and a discharge pressure of 30 bar. Using the CFD Ansys CFX commercial software, variations in both gas and oil flow were determined.

The following Von Misses stresses were obtained by applying the CFD analysis pressures: female rotor: 163 MPa, safety coefficient 5 and for male: 203 MPa, safety coefficient 4(yield limit is 800 MPa). Maximum rotors displacements are 21 μm for the female and 31 μm for the male (the clearance between the rotor is 40 μm and between the rotors and case are 100 μm).

For future research, numerical data checks will be carried out using the experimental tests carried out on the test bench of INCDT COMOTI.

ACKNOWLEDGEMENT

This work was carried out within "Nucleu" Program TURBO 2020, supported by the Romanian Minister of Research and Innovation, project number PN 16.26.06.07.

REFERENCES

[1] Bein, T.W., Hamilton, J.F., 1982. Computer modeling of an oil flooded single screw compressor. In: Proc Int Compressor Conf at Purdue. Paper 383.
[2] Boblitt, W.W., Moore, J., 1984. Computer modeling of single-screw oil flooded refrigerant compressors. In: Proc. Int. Compressor Conf. at Purdue. Paper 506.
[3] Jianhua, Wu, Guangxi, Jin, 1988. The computer simulation of oilflooded single screw compressors. In: Proc. Int. Compressor Conf. at Purdue. Paper 646.
[4] Peric, M., 1985. A Finite Volume Method for the Prediction of Three Dimensional Fluid Flow in Complex Ducts (PhD thesis). Imperial College of Science, Technology & Medicine, London.
[5] Demirdzic, I., Peric, M., 1990. Finite volume method for prediction of fluid flow in arbitrary shaped domains with moving boundaries. Int. J. Numer. Methods Fluids 10, 771.
[6] Demirdzic, I., Muzaferija, S., 1995. Numerical method for coupled fluid flow, heat transfer and stress analysis using unstructured moving mesh with cells of arbitrary topology. Comp. Methods Appl. Mech. Eng. 125, 235e255.
[7] Kovacevic, A., Stosic, N., Smith, I.K., 1999. Development of CADCFD interface for screw compressor design. In: International Conference on Compressors and their Systems, London, IMechE Proceedings, pp. 757e767. Paper C542-075
[8] Prasad, B. G. Shiva, 2004. CFD for positive displacement compressors. In: Proc. Int. Compressor Conf. at Purdue. Paper 1689.
[9] API 694 „Rotary-Type Positive-Displacement Compressors for Petroleum, Petrochemical, and Natural Gas Industries”, 4th edition, 2004;
[10] Constantin,V., Palade, V. – Organe de masini şi mecanisme, vol.I, Galaţi, Editura Fundaţiei Universitare „Dunărea de Jos”, Galaţi, 2005;
[11] N Stosic, E. Mujic and I K Smith - The Design of a High Efficiency Oil Flooded Gas Screw Compressor, City University London, November 2009;