30th Congress of the International Council of the Aeronautical Sciences
DCC, Daejeon, Korea ; September 25-30, 2016

COMPARATIVE ANALYSIS OF AIR FLOW IN THE COMPRESSOR OF THE GTD-350 ENGINE

Robert SZCZEPANIAK*, Robert BĄBEL*, Wit STRYCZNIEWICZ**
***Air Force Academy Poland, **Institute of Aviation Poland**
[1] Corresponding Author E-mail: robert.szczepaniak@o2.pl

Keywords: *GTD-350 Engine, CFD, Internal flow, Ansys, SolidWorks*

Abstract

This work illustrates the possibility of applying the numerical calculation to illustrate physical phenomena and to calculate the values of airflow characteristics in the axial compressor of the GTD-350 engine. For the purpose of the study two types of simulation software were used: Solid Works Flow Simulation [3] and Ansys Fluent [4]. A 3D model of the compressor was prepared based on the measurements in SolidWorks. This model was then tested in simulations in SolidWorks and Ansys followed by an analysis of the simulation results.

1 Introduction

Progress in the numerical and computational methods allowed to precisely model flow inside turbomachines. Thus, incorporation of the numerical simulations at the initial stage of the design process enabled to reduce the number of manufactured prototypes and shorten the design time.

Nowadays, the Computational Fluid Dynamics (CFD) is used also for optimisation of the performance of turbomachinery parts like jet engine compressors and turbines [5-12].

The main goal of the article is to examine the feasibility of methodology based on Computer Aided Engineering and reverse engineering for investigation of flow inside of compressor of popular turboshaft engine GTD-350. The analysis with use of two commercial CFD solvers was performed in order to provide comparative analysis. The results of the simulations are in good agreement with characteristic provided by the engine manufacturer. The outcome of the work proved that methodology combining reverse engineering and CFD simulations can be used for investigations of flow inside turboshaft engines modules.

2 GTD-350 Engine

Engine GTD-350 is a low-power turboprop engine (Fig 1) and constitutes a part of the twin-engine drive unit of the helicopter Mi-2. It was designed at the beginning of the 60s of the XX century by an engineer, Siergiej Piotrowicz Izotow of Klimov designing office. Starting from 1966 it was manufactured in the Mechanical Plant, WSK Rzeszów, by the agreement between Poland and Russia.

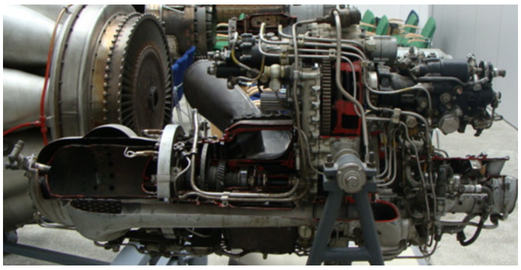

Fig. 1. GTD-350 engine

This engine is characterised by a special construction with reversed airflow. The compressor consist of axial and centrifugal compressor. The axial compressor consists of seven compression stages with stationary flow ducts. It is driven by a single-stage free turbine. The turbine consisting of two stages with stationary flow ducts drives necessary aggregates and the helicopter rotor [2].

A model of the axial compressor GTD-350 was created in SolidWorks using technical drawings, cross-sections of the blade profiles and basic measurement devices. This model consists of the inlet to the engine (casing integrated with flow ducts), compressor casing, 14 blade discs (7 rotor discs and 7 duct discs) and two pipe extensions for stabilization of the airflow.

Tab. 1 GTD-350 engine technical data [1]

Technical data	
Length	1350 mm
Width	522 mm
Height	680 mm
Mass (dry)	139 kg
Performance	
Power	294 kW
Compression ratio	6
Engine speed	45 000 rps
Temperature before the turbine	940 ^{o}C
Fuel consumption	0.496 kg/kWh 0.365 kg/KMh

The parameters of the engine given by the manufacturer are illustrated in Figure 2.

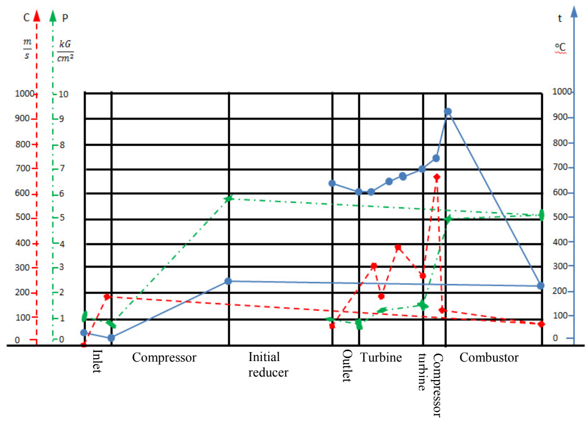

Fig. 2. Parameters of the GTD-350 engine [1]

3 Reproduction of the compressor blades

A copy of the blades was prepared using a special mould made of silicone rubber „Gumosil S" (Figure 2). Before filling the mould it was degreased and the swage was covered by Vaseline. After setting the silicone form, a few blade casts were created using FC52 resin, which is considered optimal for a thin-walled model. Subsequently, the blade curves were duplicated using these casts. The casts were specifically marked to allow precise cutting what was crucial for creating profiles.

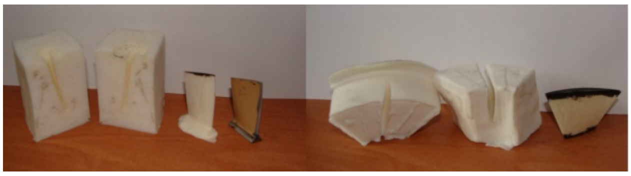

Fig. 3. Moulds and casts of the first stage of the compressor and the inlet to the engine.

The cross-section if the blades inside the moulds are shown in Figure 4. The cross-section of the blades has been scanned. High resolution of the scanner was set in order to accurately reproduce the blades geometry.

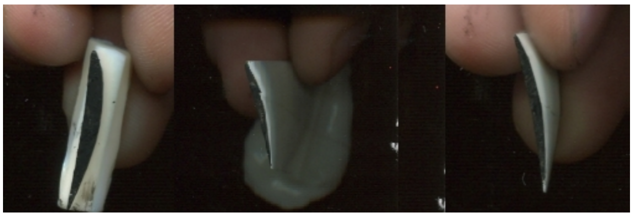

Fig. 4. The cross-section of the blades (bottom, middle and top profile of the blade)

The GTD-350 engine blades share the same geometry. The difference between the blades in the stages is the scaled size. Therefore, the geometry of one scanned blade was used for reconstruction of all stages of the compressor. Geometry of the compressor casing and the shaft has been reproduced based technical drawings and measurements of the engine. Based on this data a detailed model of the compressor was made in CAE software.

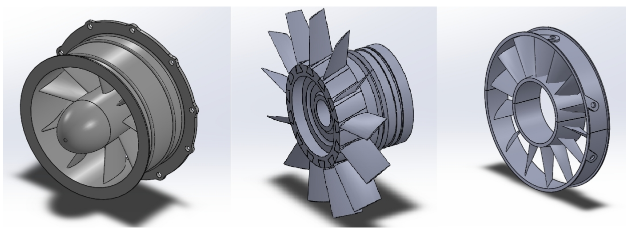

Fig. 5. CAD models of GTD-350 parts, from left: inlet, first stage of the rotor and stator

In the next step, in order to provide good quality of meshing a more coarse geometry was created. The casing has been simplify and the stages of the compressor has been separated (see Fig 6). An additional elongated computational domain has been added at the end of the compressor. It was because the centrifugal compressor has not been modelled. The elongated area was added in the CAD model in both solvers. Additionally, the elongated area was added at the front of the computational area in Ansys Fuluent.

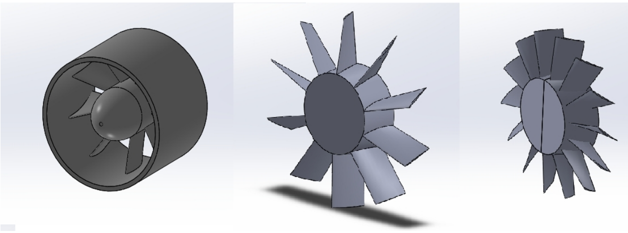

Fig. 6. Parts of the engine applied in flow simulations

The models are presented on Fig 7 (SolidWorks) and Fig 8 (Ansys). In order to reduce the computational time and set fine meshing only one blade from each stage of the compressor was modelled in the Ansys Fluent.

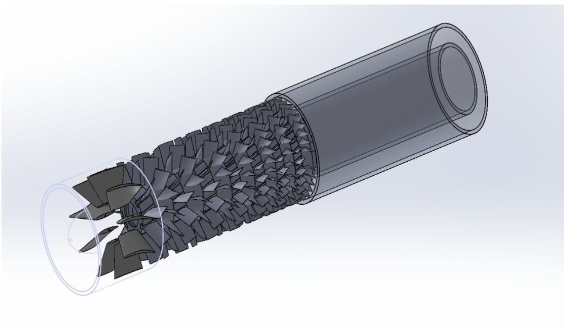

Fig. 7. CAD model of GTD-350 engine in SolidWorks

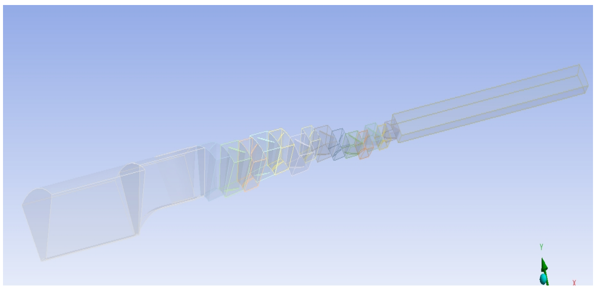

Fig. 8. CAD model of GTD-350 engine in Ansys

4 Computational simulations of flow inside GTD-350 engine axial compressor

4.1 Parameters

The following parameters were applied in the simulations:

- Compressor rotational axis congruent with axisZ,
- Rotational speed 45,000 rps to the right looking at the inlet to the engine,
- Boundary parameters: inlet – pressure 1 atm, temperature 288.15K, outlet - pressure 1 atm, temperature 288.15K.

4.2 SolidWorks Flow Simulation

The flow simulation was performed with use of *rotating regions* mode. Exemplary rotating zones are illustrated in Fig 9. The rotation speed and direction was set in the rotation zones. The parameters of meshing was set in the simulation creator.

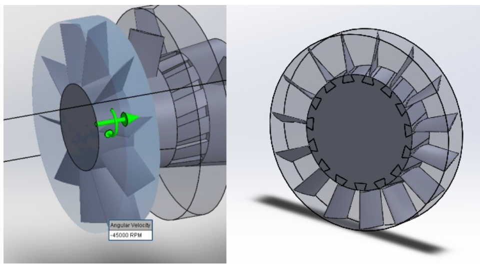

Fig. 9. Exemplary rotation zone

The result of the simulation in form of the temperature change along the compressor is shown in Fig 10. Figure 11 illustrates the temperature map in the cross-section of the compressor.

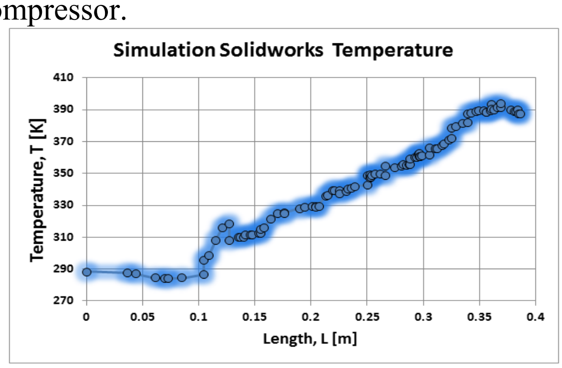

Fig. 10. Air temperature along the compressor

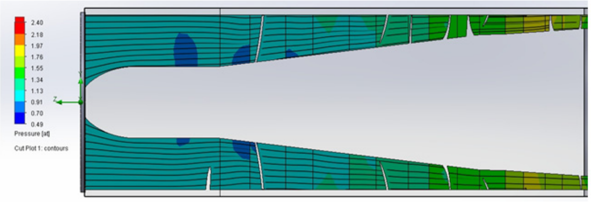

Fig. 11. Temperature field in the compressor

Pressure distribution and 2D pressure map is show in Fig 12 and 13 respectively.

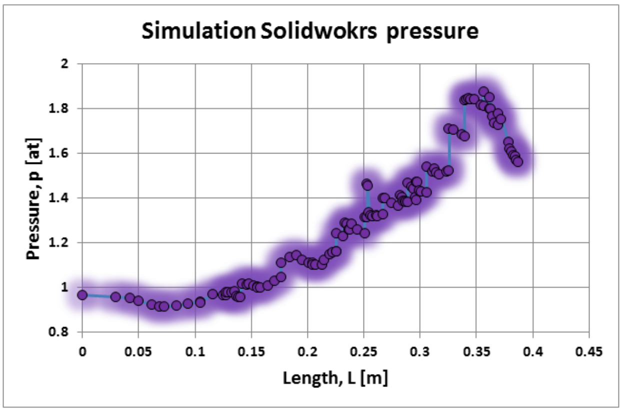

Fig. 12. Pressure along the compressor

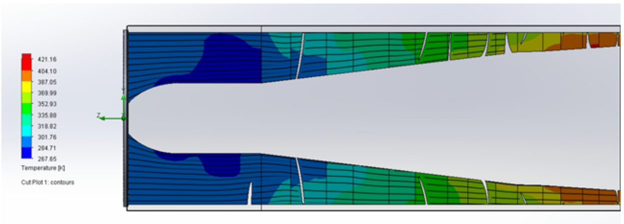

Fig. 13. Pressure field in the compressor

4.3 Ansys Fluent Simulation

The simulation was performed with use of Mixing Plane Modelling module. The module take the advantage of axisymmetric build of the compressor. This approach allows to model the turboshaft machines with use of only single blade from each stage of the compressor. Similarly, the periodicity was taken into account during meshing of the computational domain with use of Match Control. The mesh is presented in Fig 14.

The k-omega SST turbulence model hybrid initialisation was set. The results of the simulation in form of pressure distribution and vector velocity map are presented in Fig 15 and 16 respectively.

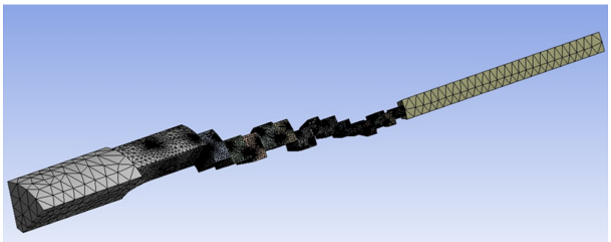

Fig. 14. Meshing of the computational domain for Ansys Fluent simulations

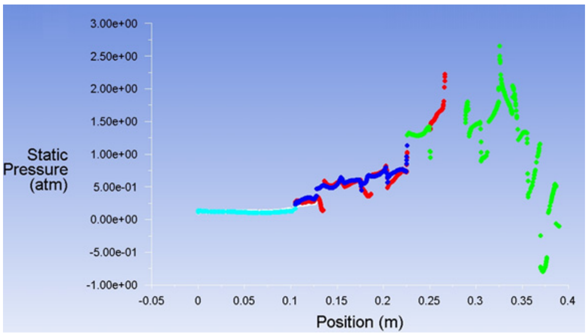

Fig. 15. Pressure distribution in the compressor

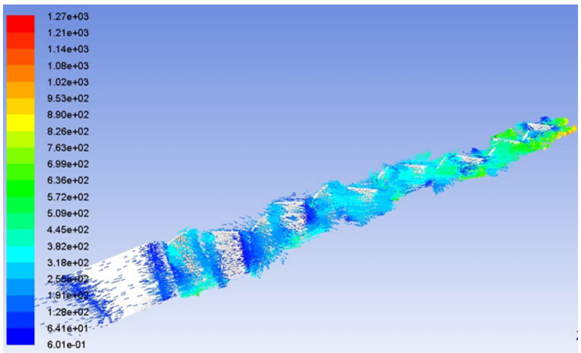

Fig. 16. Vector velocity field of the flow in the compressor

4.4 Discussion

The results of simulations in Solidworks Flow Simulation module was compared with data provided with the producer of the investigated engine. The „inlet r1" i „outlet s7" pressures was good agreement with the data form the engine manual. The temperature, pressure and the velocity distributions across the compressor was in the limits of given by the producer. Nevertheless more detailed analysis of the fluid pathlines revealed some turbulences that might be caused by not perfect reconstruction of the compressor geometry. For example a turbulent flow between rotor and stator an 5th stage of the compressor was observed. Such a flow was for

sure not intended by the engine designers and does not occurs in the engine. Similarly the pressure distribution form the Ansys Fluent simulations are not smooth with pressure drops at 6th a 7th stage of compressor (see Fig 15).
The above imperfections might be caused by errors caused by converting the CAD model from SolidWorks to Parasolid and not optimal turbulence model. The flow disturbances can be seen especially for fine meshing in Ansys Fluent.

5 Conclusion

The conclusions from the conducted experiments are following:

- to obtain credible results of calculations according to the finite element method, it is necessary to create a precise copy of the test object;
- introduction of non-uniform calculation grid (dense in the points of dynamic change of calculated parameters) is associated with acquiring broad experience;
- it is necessary also to conduct numerous calculations to gain experience in the adjustment of threshold settings and choice of turbulence models for a given phenomenon;
- the test model was created in Solid Works and therefore the obtained results of calculations are closer to the real values compared to calculations using Ansys. The reason for this are difficulties in converting from Solid Works to Ansys;
- the results from Ansys are merely an approximation at this stage of work and should be verified after using a 3D scanner to match exactly the copy of the real model in the next numerical studies;
- preliminary simulation studies are a basis for further research on methods of numerical calculations;

References

[1] *Technical description of GTD-350 engine for helicopter Mi-2* Label communications equipment, PLZ Rzeszów, 1978

[2] Gajewski T, Kolankiewicz J, *GTD-350 Engine, Basic theory, construction and operation*, Dęblin, 1978

[3] www.ansys.com

[4] www.solidworks.pl

[5] Kim S G, Lee J, Ahn Y, Lee J I, Addad Y and Ko B. CFD investigation of a centrifugal compressor derived from pump technology for supercritical carbon dioxide as a working fluid. *The Journal of Supercritical Fluids*, Vol. 86, pp 160-171, 2014

[6] Sharifi N and Boroomand M. An investigation of thermos-compressor analysis and experiment: Part 1. Validation of the numerical method. *Energy Conversion and Management,* Vol. 69, pp 217-227, 2013

[7] Chehhat A, Si-Ameur M and Boumeddane B. CFD analysis of the volute geometry effect on the turbulent air flow through the turbocharger compressor. *Energy Procedia,* Vol 33. pp 746-755, 2013

[8] Ji M, Utomo T, Woo J, Lee Y, Jeong H and Chung H. CFD investigation on the flow structure inside thermo vapor compressor. *Energy,* Vol. 35 Issue 6, pp 2694-2702, 2010

[9] Jiao K, Sun H, Li X, Wu H, Krivitzky E, Schram T and Larosiliere L M. Numerical simulation of air flow through turbocharger compressors with dual volute design. *Applied Energy,* Vol. 86, Issue 11, pp 2494–2506, 2009

[10] Ooi K T and Zhu J. Convective heat transfer in a scroll compressor chamber: a 2-D simulation. *International Journal of Thermal Sciences*, Vol. 43, Issue 7, pp 677-688, 2004

[11] Benini E and Biollo R. Aerodynamics of swept and leaned transonic compressor-rotors. *Applied Energy*, Vol. 84, Issue 10, pp 1012–1027, 2007

[12] Borello D, Hanjalic K and Rispoli F. Computation of tip-leakage flow in a linear compressor cascade with a second-moment turbulence closure. *International Journal of Heat and Fluid Flow*, Including Special Issue of Conference on Modelling Fluid Flow (CMFF'06), Budapest, Vol. 28, Issue 4, pp 587–601, 2007

Copyright Statement