THERMAL MANAGEMENT OF HELICOPTER SUB-SYSTEM IN VARIOUS STATES OF FLIGHT

Katarzyna Surmacz, katarzyna.surmacz@ilot.edu.pl, Łukasiewicz Research Network - Institute of Aviation (Poland)

Adam Dziubiński, adam.dziubinski@ilot.edu.pl, Łukasiewicz Research Network - Institute of Aviation (Poland)

Frieder Hirsch, frieder.hirsch@koptergroup.de, Kopter Germany (Germany)

Abstract

The paper describes the aerothermodynamic analysis of the conceptual design of the modern helicopter SH09 developed by the Kopter Group AG. For better understanding of the cooling process in the helicopter, Computational Fluid Dynamics tools were used. The object of research was the internal flow in the space under the cowl of a helicopter. The simulation also takes into account the external influence, including the downwash from the main rotor of a helicopter. The CFD analysis of the SH09 helicopter concerned different states of flight and their impact on the cooling process. The results of the helicopter operating in several conditions such as: hover, forward, rearward and lateral flight are presented. The aim of the computational investigation was to find out key parameters of cooling process of the helicopter, in particular: the main directions of flow in different conditions of a helicopter flight, the airflow cooling under normal and extreme conditions, the impact of the flight conditions, ambient conditions and oil cooler fan setting, the efficiency of the cooling system.

1. INTRODUCTION

1.1. Background

The most significant advantage of a helicopter is the ability to perform a flight in each direction (forward, rearward, lateral), at low speed and the ability for hovering flight. One of the factors that may affect the limitation of the scope of applicability of a rotorcraft is the performance of the cooling near its engine. Depending on the flight conditions (flight direction, altitude, ambient temperature) heat exchange with the surroundings may be limited. Therefore, the aerothermodynamic analysis is performed at the helicopter design stage. At this stage, the complex aerothermodynamic processes connected with heat conduction and fluid flow can be analyzed using computational methods.

1.2. Purpose of research

The paper describes the CFD flow analysis under the forward and middle cowl of the SH09 helicopter. CFD calculations were performed to determine the following issues: the main flow directions, pressure levels and distribution under the cowl section, temperature levels on the selected components in the internal section, the efficiency of the airflow cooling under normal and extreme conditions (for example "hot and high" atmospheric conditions), the efficiency of the fresh air supply and cooling performance. The impact of helicopter flight conditions and ambient conditions on the cooling process was also investigated.

2. METHODOLOGY

2.1. Object of research

The paper describes the CFD analysis of the conceptual design of the modern helicopter SH09 (an image of the SH09 helicopter is presented in the Fig. 1.) offered and developed by Kopter Group AG. The SH09 helicopter is positioned in the 2.5 metric ton class (maximum take-off weight MTOW with internal load 2650 kg) and reaches the cruise speed (at MTOW) of 140 kts, the maximum range 800 km and the maximum endurance from 3.7 hrs to 5 hrs (http://www.koptergroup.com [4]).

Fig.1. An image of the SH09 helicopter http://koptergroup.com/gallery/13

Presented at 45th European Rotorcraft Forum, Warsaw, Poland, 17-20 September, 2019

2.1.1. Geometric model

The object of research was the space under the cowl of a SH09 helicopter. The computational geometry consisted of internal and external parts. As the internal flow strongly depends on local pressure and speed at all openings the external flow was also simulated. External part of model included the basic external shape of the helicopter (as shown in the Fig.2.), such as fuselage, tail boom, vertical and horizontal stabilizers, mast. The main rotor was simulated as a FAN model (with a constant pressure jump on the disc).

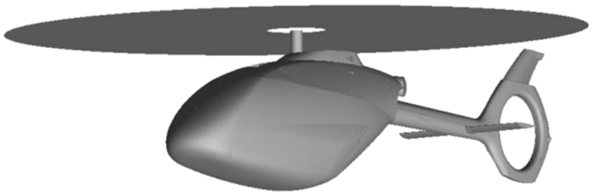

Fig.2. A model of the SH09 helicopter

The internal part (Fig.3.) was placed between lower deck and forward, middle and aft cowl and was the main object of the aerothermodynamic analysis. Computational internal domain includes the main components which can have an impact on the temperature distribution and on the air flow in the space under the cowl. The main components necessary from the heat transfer point of view are: environmental control system (ECS), main gearbox, hydraulics, engine firewall, pitch control system, oil cooler, engine exhaust. The engine compartment was not analyzed.

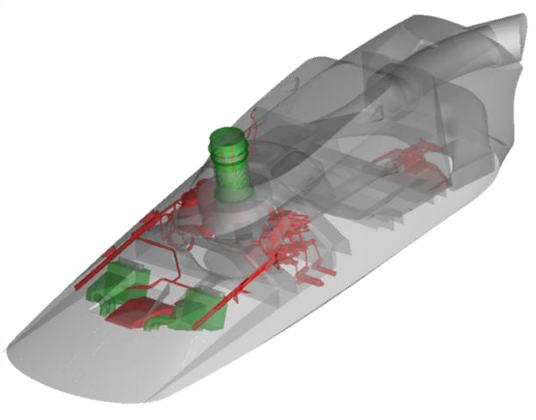

Fig.3. The internal part of a geometric model

2.1.2. Computational meshes

A preparation of the CAD model for the CFD calculations was made using tools of the ANSYS FLUENT package [1]. Computational grids were made using ICEM CFD [2].To simulate a flow around the SH09 helicopter two computational grids were generated: internal and external. All model's geometry modifications and computational grids have been made using ICEM CFD code. The cubical domain of dimensions: 120x120x120 [m] was generated for the external part. The domain was generated as an unstructured grid of tetrahedral cells (4 M cells). To model the external part a relatively fine mesh (13 M cells) was generated using tetrahedral elements and several layers of prisms as a boundary layer. Internal and external grids were merged in ANSYS FLUENT software using the "interface" boundary condition. Connections between external and internal models are marked in red in the Fig.4. Some details of grid are shown in the Fig.5.

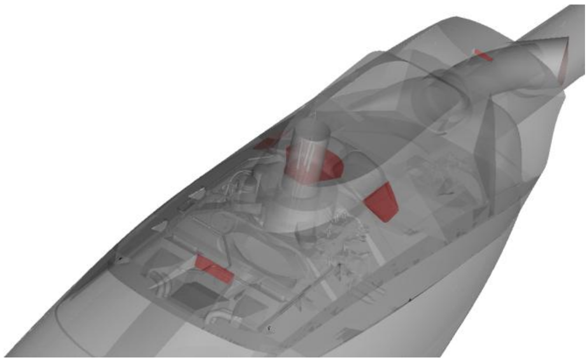

Fig.4. Connections between external and internal models (marked in red)

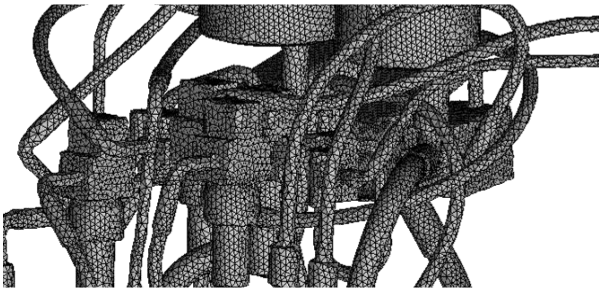

Fig.5. Details of hydraulic system discretization

2.2. Research tools

The numerical analysis was carried out using Computational Fluid Dynamics tools. Simulations were performed using ANSYS FLUENT code [2], widely recognized as industry standard. This numerical algorithm is based on the finite-volume method. The software allows the study of compressible or incompressible, viscous or non-viscous, steady or unsteady flows. In the applied code many turbulence models have been implemented (i.a. Spalart-Allmaras (1 eqn.), k-epsilon (2 eqn.), k-omega (2 eqn.), transition SST (4 eqn.)).

Presented at 45th European Rotorcraft Forum, Warsaw, Poland, 17-20 September, 2019

The main goal of this study was an aerothermodynamic analysis of the flow inside the nacelle and outside of the fuselage of a helicopter. Computational internal domain includes the main components which can have an impact on the temperature distribution and air flow in a section under the cowl. Specific surfaces in the inner section have been modelled as virtual thickness surfaces, which gives an opportunity to calculate the heat fluxes. The steady state calculation is assumed to be adequate to prove that the design can sustain in temperatures that are sufficient for cover materials to keep the necessary properties.

The geometric model of the cooling section of a SH09 helicopter was supplied in STEP format. The initial CAD model was very detailed, so many parts had to be simplified or removed. A preparation of the CAD model for the CFD calculations was made using the SPACECLAIM tool [3] and ICEM CFD software that are a part of the ANSYS FLUENT package. The first program was used for initial cleaning and simplification process, the second one to produce geometric models and mesh generation.

2.2.1. Computational models

Research was carried out based on Reynolds-Averaged Navier-Stokes approach (RANS) implementing k-ε RNG turbulence model.

The following assumptions and settings were used in the study:

- three-dimensional steady calculations
- ideal gas
- compressible flow
- a main rotor pressure jump Δp=268 Pa (helicopter mass: 2650 kg).

Some of the settings were taken from the paper [5].

2.3. Schedule of calculations

The CFD analysis of the SH09 helicopter concerned different states of flight and their impact on the cooling and ventilation process. The numerical analysis of the flow around the SH09 helicopter and inside its cowl section was performed for five flight conditions. Computational cases included different helicopter states of flight such as: hover, fast forward flight (with the speed 77.7 m/s), rearward flight (with the speed 18 m/s) and lateral flight to the left (with the speed 18 m/s). Most of the calculations were performed in so-called "high & hot" conditions (high temperature - about 40 [°C] and high altitude - about 2000 [m]).The aim of calculations was to examine the effectiveness of the cooling system in difficult atmospheric conditions.

3. RESULTS OF RESEARCH

3.1. Results of calculations

The purpose of the computational investigation was the aerothermodynamic analysis of the flow inside the nacelle and outside of the fuselage of a helicopter and to find out key parameters of cooling process. The following issues were examined in particular: the main directions of flow in different conditions of a helicopter flight, the airflow cooling under normal and extreme conditions, the impact of the flight conditions, ambient conditions and oil cooler fans setting, the efficiency of the cooling system (including oil cooler). The results of this investigation allowed obtaining valuable and useful information (both quantitative and qualitative). Based on the performed simulations the following was determined: values of mass flow rate (MFR) for selected inlets and outlets, the temperature distribution in selected cross-sections and on the surfaces, the pressure distribution in vertical and horizontal cross-sections, the velocity distribution in selected cross-sections, the velocity pathlines and vectors showing the airflow direction. A comparison of results obtained for different flight conditions and different ambient conditions is presented.

3.1.1. Analysis of flow direction

The paper presents selected results in the form of images illustrating the directions of flow under the cowl (subsection 3.1.1), and the temperature distribution in the selected cross-section illustrating the cooling and heating process of the structures under the cowl (subsection 3.1.2) of the SH09 helicopter. The drawings can be compared qualitatively, because for all test cases the same ranges of velocity and temperature are used (the color blue represents low values; the red color represents high values).

Figures 6-10 show pathlines colored by velocity magnitude (colorful lines) released from the surface inlet at the front of the helicopter and from fans of the oil cooler. In addition, the velocity vectors (in the form black arrows) in the selected cross-section of the chamber are presented.

Presented at 45th European Rotorcraft Forum, Warsaw, Poland, 17-20 September, 2019

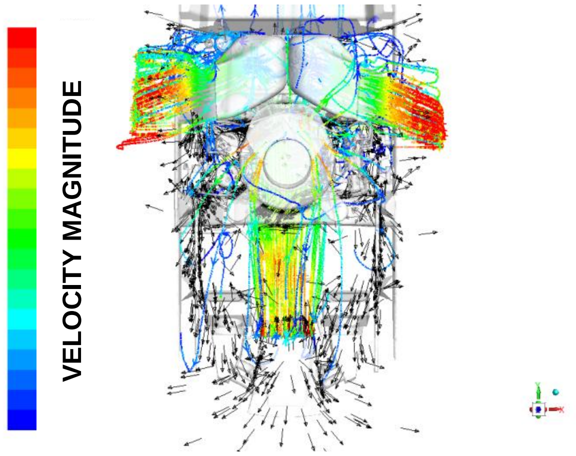

Fig.6. Visualization of the flow under the cowl, hover in standard conditions

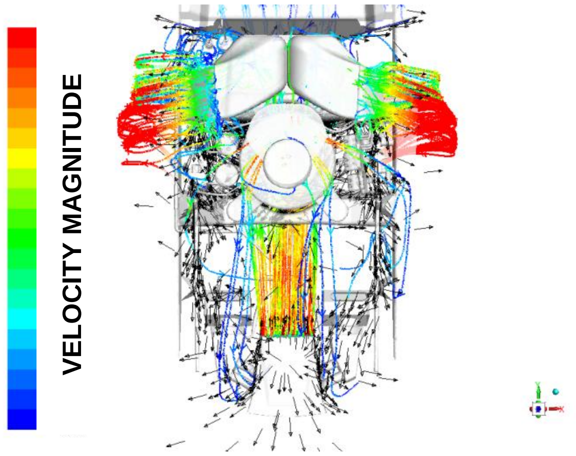

Fig.7. Visualization of the flow under the cowl, hover in "hot&high" conditions

As can be seen from the illustrations, for hover and forward flight cases, a lot of fresh air flows into the zone under the cowl and provides good cooling conditions. The oil cooler fans also work efficiently and generate a high air flow rate through the cooler.

In the analysis of mass flow rate through the main inlet to the section under the cowl and the inlet and outlet of the oil cooler, it was found that during the rearward flight in the "hot & high" conditions, the effective flow rate is relatively small.

However, in the case of lateral flight in "hot & high" conditions, the work of one of the fans is disturbed by the side wind, while the other works effectively.

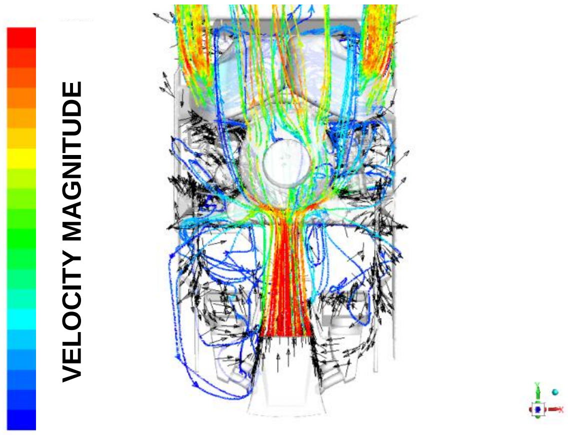

Fig.8. Visualization of the flow under the cowl, fast forward flight in "hot&high" conditions

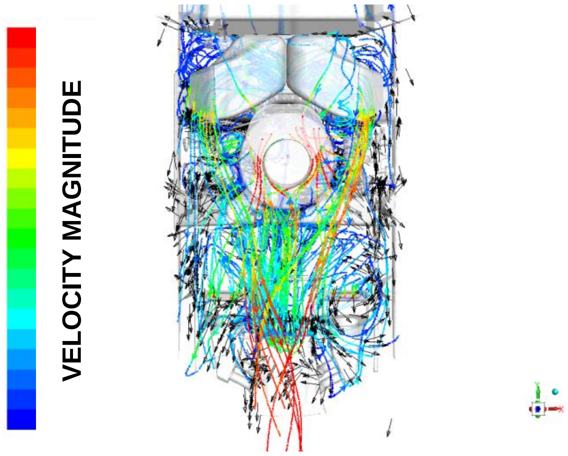

Fig.9. Visualization of the flow under the cowl, rearward flight in "hot&high" conditions

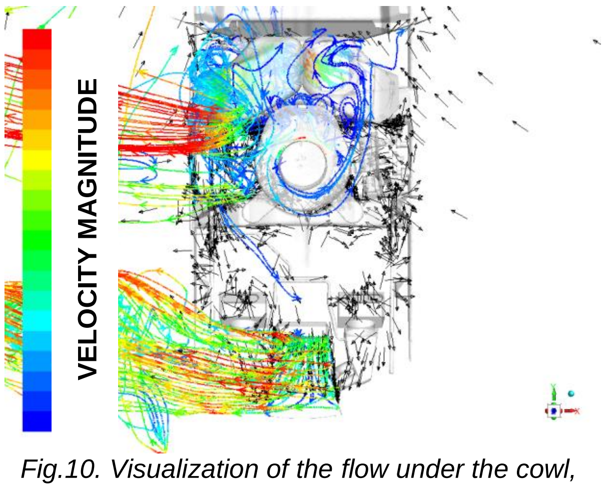

Fig.10. Visualization of the flow under the cowl, lateral flight in "hot&high" conditions

3.1.2. Analysis of temperature distribution

The results of temperature distributions in several vertical and horizontal cross-sections are presented below. It shows the temperature distributions around important devices as main gearbox, hydraulics with tubing, pitch control

Presented at 45th European Rotorcraft Forum, Warsaw, Poland, 17-20 September, 2019

system, oil cooler, and environmental control system (ECS). For the first calculation case, at standard conditions (temperature of 15 [°C] and altitude of 500 [m]), a low temperature level was observed. Elements of structures under the cowl are not heated to high temperatures. The cooling system plays a significant role in "hot & high" conditions. In cases of hover and fast forward flight at these conditions, the efficiency of the cooling system is very good, only a slight increase in temperature was observed at the rear part of the cowling (near the engine exhaust).

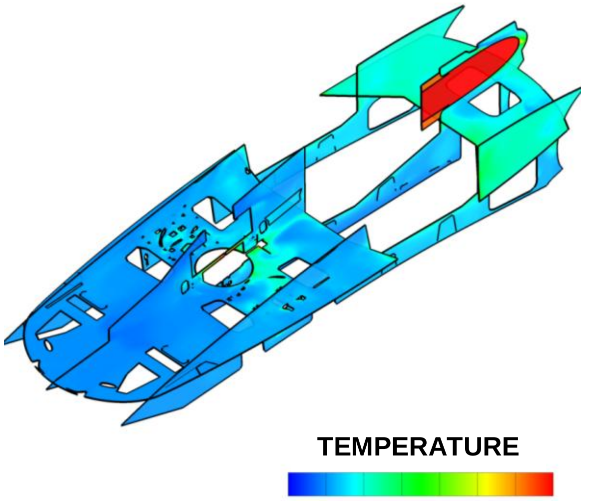

Fig.11. Temperature distribution in cross sections, hover in standard conditions

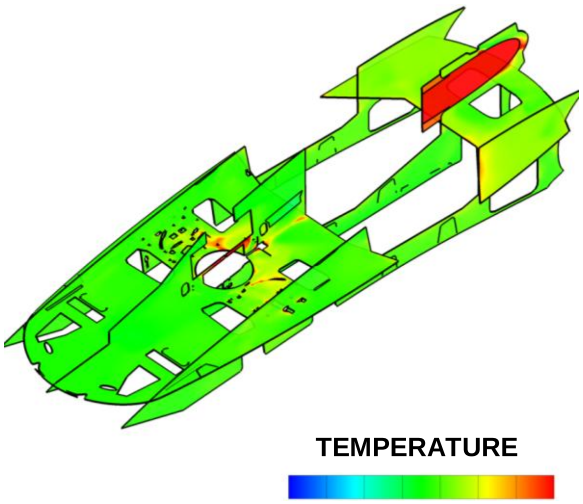

Fig.12. Temperature distribution in cross sections, hover in "hot&high" conditions

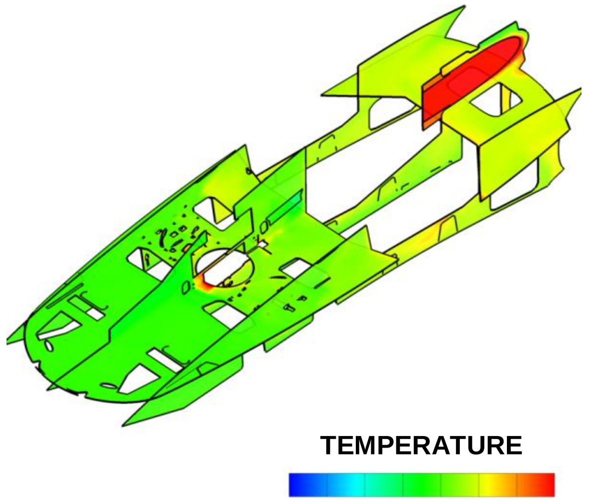

Fig.13. Temperature distribution in cross sections, fast forward flight in "hot&high" conditions

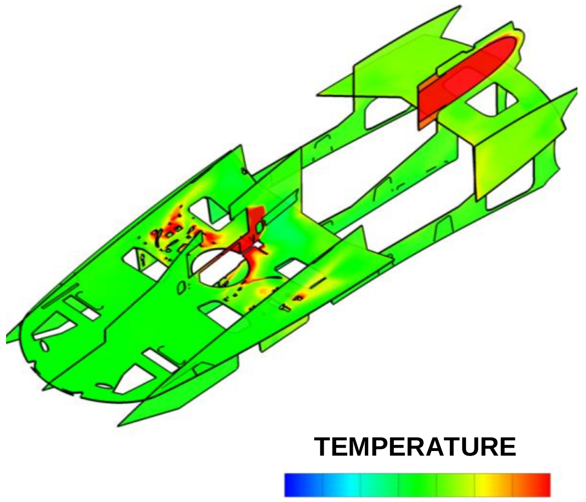

Fig.14. Temperature distribution in cross sections, rearward flight in "hot&high" conditions

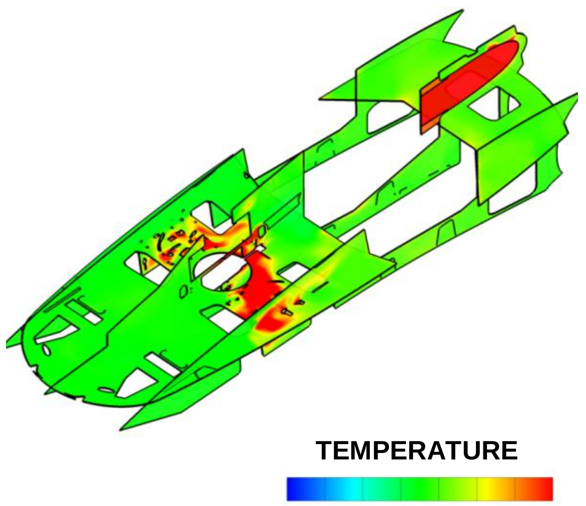

Fig.15. Temperature distribution in cross sections, lateral flight in "hot&high" conditions

Presented at 45th European Rotorcraft Forum, Warsaw, Poland, 17-20 September, 2019

In cases of rearward and lateral flights, heating of components such as: main gearbox, pitch control system and hydraulics was noticed. The highest temperature levels (marked in red) were observed on the surfaces of these devices and in their vicinity. The most critical maneuver in terms of cooling is a lateral flight of a helicopter in "hot & high" conditions.

4. RESULTS OF RESEARCH

On the basis of the results obtained using numerical methods it was possible to determine: states of flight of the SH09 helicopter with good or insufficient cooling and the main potential for improvement. The results confirm that the design can keep temperatures on a level, which allows the sub-systems to operate properly. Performed simulations showed that in most cases the cooling process is sufficient and acceptable. An idea to improve the efficiency of the cooling process is the use of additional air inlets on the side of the nacelle (for example NACA inlets). Another idea is to change the shape of the outlet pipes of the oil cooler or to move the outlets of the oil cooler to benefit from the suction effect due to external flow.

5. BIBLIOGRAPHIES

[1] ANSYS, "ANSYS Fluent 14.5 User's guide"
[2] ANSYS, "ANSYS ICEM CFD User's guide"
[3] ANSYS SpaceClaim User's guide 2016
[4] http://www.koptergroup.com
[5] "Simplified thermo-fluid model of an engine cowling in a small airplane" Łapka P., Seredyński M., Furmański P., Dziubiński A, Banaszek J., Aircraft Engineering and Aerospace Technology, ISSN: 0002-2667, 2014

Presented at 45th European Rotorcraft Forum, Warsaw, Poland, 17-20 September, 2019