Strength analysis of the rotor hub of an unmanned helicopter

K Cygnarowska\textsuperscript{1}, Z Czyż\textsuperscript{1}, P Karpiński\textsuperscript{2} and K Skiba\textsuperscript{2}

\textsuperscript{1}Military University of Aviation, Aeronautics Faculty, 35 Dywizjonu 303 St., 08-521 Dęblin, Poland
\textsuperscript{2}Lublin University of Technology, Faculty of Mechanical Engineering, 36 Nadbystrzycka St., 20-618 Lublin, Poland

z.czyz@law.mil.pl

Abstract. This article presents a method of strength analysis of the rotorcraft component which is the rotor hub of an unmanned helicopter with a MTOW up to 150 kg. The simulation takes into account two materials of which individual elements are made, i.e. steel and aluminum alloy. The load scheme and element support were defined, and numerical calculations were performed in the Ansys Workbench software. The simulation results were presented in the form of stress maps and were subjected to detailed analysis for individual elements. In the last stage, critical places in the assessment of the strength of the elements were indicated. On the basis of the performed strength analysis, the obtained results were individually interpreted for each of the tested elements depending on the adopted safety factors.

1. Introduction
The aviation industry is a constantly developing branch of the economy. Therefore, numerous studies are carried out to develop and optimise new aircraft designs. A particularly important issue in the study of aircraft is the optimisation of their design in terms of reducing aerodynamic drag, increasing lift and minimising the weight [1-3]. The use of numerical methods in the design process significantly reduces the time of product development and costs compared to experimental methods. In recent years, unmanned aerial vehicles have gained in importance, hence many scientific works focus on this subject [4-8]. In this work, the rotor hub of the unmanned aerial vehicle was analysed. The main rotor hub is a key element in the rotorcraft structure. The rotor blades are attached to it using joints. The rotor is driven by the rotation of the shaft which transmits the torque from the engine. In addition, the rotor hub transmits the lift from the blades to the fuselage of the rotorcraft. This subassembly is mounted on a shaft and is therefore affected by inertia forces, aerodynamic forces and moments. The analysis of the aerodynamic effects on the main rotor blades or the hub rotor itself can be performed using a wind tunnel [9-13] or based on numerical CFD analyses (Computational Fluid Dynamics) [14]. CFD analyses in the field of aircraft make it possible to study their stability [15], aerodynamic effects occurring on aircraft components such as a stabilizer [16], fuselage [17], rotor [18] or occurring on the entire structure of the aircraft [19].

CFD analyses are extended by coupled analyses that enable the analysis of a coupled problem by dividing the system into different subsystems. Then, separate modelling and simulation processes are performed for these subsystems, and the interaction between them is synchronised at the defined discrete points. An example of such an approach is the combination of Multi-Body Dynamics modelling with CFD research [20].
The forces and moments on the rotor hub vary depending on the phase of flight. For this reason, the rotor hub should show high resistance to complex loads and high stiffness. There are four main types of helicopter rotor hubs. These are teetering, articulated, hingeless and bearingless designs [21].

To test the strength of the element, it is necessary to carry out strength tests to determine the behaviour of the structure (deformation or destruction) due to stresses in the material. To ensure adequate strength, it is necessary to choose the right material. Aluminum alloys play a special role in aviation due to their favorable strength-to-weight ratio [22].

The basic methods of strength testing are experimental, analytical and numerical methods. The works [23] and [24] present the results of the helicopter rotor hub fatigue test. The method for the detection and avoidance of static load limits on the main rotor hub [25] was also investigated. In the event of a failure of the main rotor hub, it is also possible to analyse the material structure [26].

Numerical methods are used for structures with a complex load state where obtaining results (stresses, strains, displacements) with analytical methods is highly complicated. The numerical method used to test the strength of aircraft components is the FEM method [27-29]. This method is favourable for detection of critical locations [30]. It consists in dividing the surface into finite elements that average the physical properties of the test object. Then, calculations are performed for the nodes of this division only, and the analysed physical properties are approximated on the basis of solutions for the nearest nodes [31]. The research of the helicopter rotor hub using this method is presented in [32]. Numerical FEM tests can also be used to analyse the fatigue life prediction of a helicopter main rotor blade [33].

2. Geometric model

The preliminary design of the rotor hub unit should be subjected to strength analysis. The implementation of the CAD model into the CAE environment requires the use of model simplifications and the appropriate definition of the external force system in accordance with the real system. The rotor hub (Figure 1) is a kinematic chain of mechanisms with one degree of freedom. It aims to transfer the torque from the main gearbox to the blades of the main rotor. In addition, it transfers the lift force from the blades to the fuselage of an unmanned helicopter - in the example under consideration - with a take-off mass of up to 150 kg. The third task is to change the blade angle of attack.

In the beginning, an analysis of the model was conducted to remove collisions of the elements in the model, the possibility of simplifying the geometry and the possibility of minimising the number of elements. Then the model was implemented in the CAE programme.

In the design stage, two materials were assigned to the individual elements. The first one is a low-alloy structural steel grade 42CrMo4 (40HM) which undergoes heat treatment [34]. The second is an aluminium alloy for the plastic processing AlZn6Mg2Cu (PA9 7075). Figure 2 shows the hub elements made of the materials listed above. Table 1 shows the assumed mechanical properties of the materials used to make the hub.

| Material        | 42CrMo4 | AlZn6Mg2Cu | Unit      |
|-----------------|---------|------------|-----------|
| Young’s module  | 210     | 64         | GPa       |
| Tensile strength Rm | 1030    | 386        | MPa       |
| Yield strength Re | 880     | 303        | MPa       |
| Permissible stresses $k_r$ | 430     | 148        | MPa       |
| Permissible stresses $k_{ij}$ | 165     | 57         | MPa       |
| Permissible stresses $k_{ic}$ | 95      | 33         | MPa       |

The discretisation of the hub elements was done with tetra-type elements. The thickness of the grid was selected for each element, depending on its dimensions. Figure 3 shows the view of the shaft after the discretisation. Figures 4 and 5 show the selected key elements of the structure under investigation.
**Figure 1.** Geometrical model of the investigated rotor hub.

**Figure 2.** Hub elements made of 42CrMo4 steel (a) and AlZn6Mg2Cu aluminium alloy (b).

**Figure 3.** The shaft after the discretisation.
3. Results and analysis

The main purpose of the rotor hub is to transfer the torque from the engine to the rotor blades and the lift force from the blades to the helicopter support structure. The design assumptions are based on a helicopter's 30 kW engine. The rotor speed is defined as 1800 rpm. Based on the relationship between the engine power (P), rotor speed (n) and torque (Mo), the torque was calculated in the rotor hub. This was 159 Nm. A take-off mass of 150 kg was assumed. Therefore, a force of 1500 N was applied to the rotor shaft. The calculated centrifugal force was 14922.9 N.

The analysis of the simulation results started with the analysis of the displacement map for the hub. The maximum displacement was 0.16 mm. These displacements are the sum of axial and angular displacements of all elements. The largest displacements occurred at the end of the shaft. This is due to both the displacement of all elements along the gravitational force and the angular displacement of the rotor head. Angular displacement resulting from joint deformations is visible as radial strips along the shaft. Stress analysis is carried out on the basis of the von Misses Huber hypothesis in which all stresses are comparable to tensile stresses. Stress values were analysed for each individual component. The article presents only selected elements.
The first simulated element was the rotor hub shaft (Figure 6). The maximum stresses occurred at the point the torque was applied. This is due to the small contact area, but their value (44 MPa) is safe for the structure. The part of the shaft between the bore and the splines was the most stressed as it is the torque-loaded section. The stresses in this zone did not exceed 19 MPa. The zone loaded with the longitudinal tensile force is minimally loaded at the bottom of the shaft (stresses do not exceed 5 MPa). In the upper part, due to the thinner wall, the stresses did not exceed 10 MPa. The spline transmits the torque only in the initial zone, which indicates the possibility of shortening its length. The second element subjected to the stress analysis was the rotor hub head. On the stress map (Figure 7), it can be seen that the maximum stresses occurred at the cut-out point of the head for the joint seat and amounted to 83.6 MPa. The zones of increased stresses occurred on the cross-sections of the joint pin fixtures, which results from stretching under the influence of centrifugal force. The stresses in this zone did not exceed 67 MPa. The next analysed element was the joint (Figure 8). The maximum stresses were 108.8 MPa. The joint was loaded in the area between the tilt axis bore and the yaw axis bore where the element is stretched. The stresses exceed the allowable stress $\sigma_{uv}$ at the mounting point of the yaw axis of the rotor blade. Moreover, the stresses on the side wall between the vertical holes and the horizontal hole exceed the allowable stress $k_{max}$, and their value reaches 47 MPa. The cross-section shows the place where the stresses exceed the allowable stress $k_{cc}$, which may lead to fracture.

The next tested elements were the rotor blade grip covers (Figure 9). The maximum stresses were 69.1 MPa and exceeded the allowable stresses $\sigma_{uv}$. They are located on the radius connecting the rib with the contact surface. They occur locally and can lead to a fracture at this point. In the remaining zones, the stresses did not exceed 33 MPa.
The next elements considered were the rotor blade grips (Figure 10). The greatest stresses occurred in the vicinity of the rotor blade mounting holes. These stresses exceeded both the allowable stresses $t_{mv}$.
and \( t_{uv} \). Their value is 97.8 MPa. They result from the complex state of stresses (stretching and bending) as well as from the weakening of the cross-section through the hole. They have a local character in the cross-section of the first bore, while in the cross-section of the second bore, they do not exceed 33 MPa. This can break the blade grip at this point. Increased stresses also occurred in the area of the rotor blade connecting the blade grip with its cover. The stress value at this point did not exceed 50 MPa. The remaining elements that were analysed are characterised in the Conclusions section.

4. Conclusions
The strength analysis shows that depending on the accepted safety factors (\( t_{mv} \), \( t_{uv} \), \( t_u \)) the strength of each element can be interpreted individually. The simplest interpretation is for the elements in which the permissible stresses \( t_{mv} \) are not exceeded. These parts have been designed correctly and do not require any changes. This group includes the shaft, hub head, rotor blade swinging axis and rotor blade turning axis. Although the shaft construction seems oversized, it is correct. This is due to the use of the hole inside the shaft for the supply of hydraulic and measuring systems. The only element that may raise objections is the spline. The stresses on it show its operation in the initial section only. Therefore, its shortening should be considered. The calculation of the hub head has shown the design correctness in terms of strength. Alternatively, wall thickness and spline length optimisation can be considered.

The interpretation of the elements in which the stresses are in the range between \( t_{mv} \) and \( t_{uv} \) raises some doubts about the necessity of changes. Depending on individual cases, it is necessary to decide on structural changes or not. This group includes the rotor blade swing axis and the rotor blade twist axis. The rotor blade swing axis obtains the maximum stress on its cutting sections. Due to exceeding the admissible stresses, it is recommended to change the diameter of the workpiece for the diameter of the interaction. The stresses occurring at the point of interaction between the rotor blade and the swinging axis require a redesigning of this connection so that the axis does not break off. In a larger pivot diameter, the stress is low. Due to the bearing used, a change in diameter is not recommended. Instead, an axle bore can be made to reduce the weight of the component.

Another group consists of the elements in which the maximum stresses do not exceed the permissible \( t_s \), while the permissible \( t_{uv} \) is exceeded. Local changes in these parts are necessary, but usually there is no need to change the whole element. This group includes the hinge, the rotor blade deflection axis, the rotor blade cover, the rotor blade mounting yoke, the vibration damper holder and the rotor blade angle change holder.
The next group are the parts in which the maximum stresses are in the range between the yield strength Re and the allowable stresses t. This state of stress leads to large structural changes within the component. Stress in this range can lead to rapid fatigue wear of the component and requires advanced fatigue research.

The last group are the elements in which the maximum stress is higher than the yield point Re. The elements which are in this range require thorough design changes because the material above the yield point is not usable (its shape changes permanently). Furthermore, in the simulation of displacement and stress for such an element, the elastic material is not reliable. In the last two groups, there are none of the elements under consideration. This indicates the correct design of the rotor hub.

Acknowledgments
This work has been financed by the National Centre for Research and Development under the LIDER program. Grant Agreement No. LIDER/45/0177/L-9/17/NCBR/2018.

References
[1] Lyu Z and Martins J 2014 J. Aircr. 51(5) 1604-1617
[2] Kenway G and Martins J 2014 J. Aircr. 51(1) 144-160
[3] Zhu J, Zhang W and Xia L 2016 Arch. Computat. Methods Eng. 23(4) 595-622
[4] Kontogiannis S and Ekaternakis J 2013 Aerosp. Sci. Technol. 29(1) 339-350
[5] Ozdemir U, Aktas Y, Vuruskan A, Dereli Y, Tarhan A, Demirbag K, Erdem A, Kalaycioglu G, Ozkol I and Inalhan G 2014 J. Intell. Robot. Syst. 74 371-393
[6] Panagiotou P and Yakinthos K 2020 Aerosp. Sci. Technol. 99 105575
[7] Czyż Z and Wendeker M 2020 Sensors 20(12) 3360
[8] Czyż Z, Karpiński P and Stryczniowicz W 2020 Sensors 20(19) 5537
[9] Reich D, Elbing B, Bereznin C and Schmitz S 2014 J. Am. Helicopter Soc. 59(3) 1-12
[10] Reich D, Willits S and Schmitz S 2017 J. Aircr. 54(5) 1692-1704
[11] Reich D, Sinding K and Schmitz S 2018 Exp. Fluids 59(7) 116
[12] Siadkowska K 2020 IEEE 7th International Workshop on Metrology for AeroSpace - MetroAeroSpace (Pisa, Italy) 610-614
[13] Czyż Z and Siadkowska K 2020 IEEE 7th International Workshop on Metrology for AeroSpace - MetroAeroSpace (Pisa, Italy) 625-629
[14] Lim J and Dimanlig A 2010 36th European Rotorcraft Forum (Paris, France) 051
[15] Czyż Z, Łusiak T, Karpiński P and Czarnigowski J 2018 J. Phys. Conf. Ser. 1101(1) 012003
[16] Czyż Z and Karpiński P 2020 Aviation 23(4) 114-122
[17] Kusyumov A, Mikhailov S, Garipov A, Nikolaev E, Barakos G 2012 Transaction on Control and Mechanical Systems 1(7) 318-324
[18] Datta A, Nixon M and Chopra I 2007 J. Am. Helicopter Soc. 52(4) 287-317
[19] Biava M, Khier W and Vigevano L 2012 Aerosp. Sci. Technol. 19(1) 3-18
[20] Siadkowska K, Raczyński R and Wendeker M 2019 IOP Conf. Ser.: Mater. Sci. Eng. 710 012009
[21] Leishman J G 2006 Principles of helicopter aerodynamics 2nd ed. (Cambridge: Cambridge University Press) chapter 4 172-174
[22] Dursun T and Soutis C 2014 Mater. Des. 56 862-871
[23] Giglio M, Manes A and Viganò F 2011 Fatigue Fract. Engng Mater. Struct. 35, 317-327
[24] Viganò F, Manes A, Giglio M and Mariani U 2011 Eng. Fail. Anal. 18(1) 97-109
[25] Horn J and Sahani N 2004 J. Aircr. 41(2) 372-379
[26] Wanhill R 2003 Practical Failure Analysis 3 59-69
[27] Oskuei R, Keikhosravry M and Soutis C 2010 Aeronaut. J. 114 1156
[28] Ostergaard M, Ibbotson A, Le Roux O and Prior A 2011 Aeronaut. J. 1 83-103
[29] Grbovic A and Rasuo B 2012 Engineering Failure Analysis 26 50-64
[30] Maksimović S 2005 Sci. Tech. Rev. LV(1) 15-22
[31] Huebner K, Dewhirst D, Smith D and Byrom T 2001 *The finite element method for engineers 4th ed.* (John Wiley & Sons, Inc.) chapter 1 3-16

[32] Skiba K 2019 *IOP Conf. Ser.: Mater. Sci. Eng.* **710** 012003

[33] Shahani A and Mohammadi S 2016 *Meccanica* **51** 1869-1886

[34] Szala M, Winiarski G, Wójcik Ł and Bulzak T 2020 *Materials* **13**(9) 2022