Strength analysis of a propulsion shaft dedicated for the main rotor test bench

K Skiba, R Raczyński, R Kliza and M Wendeker

Lublin University of Technology, Faculty of Mechanical Engineering, Department of Thermodynamics, Fluid Mechanics and Aviation Propulsion Systems, Nadbystrzycka 36, 20-618 Lublin, Poland
k.skiba@pollub.pl

Abstract. This article presents the strength analysis of various drive shaft concepts for the test stand for research of aircraft's main rotors. The results of stress analysis for conceptual shaft models are presented. A technological solution was developed on the basis of the obtained results to meet the design requirements and reduce the impact of maximum stress to a minimum. The aim of the study was to develop a design solution for a propulsion shaft research allowing for the most optimal way of conducting the research installation from a stationary to a rotary system (data and power collector system).

1. Introduction

Flight permits for new aircraft are granted after a number of research and analyses [1-3]. The implementation of modifications and upgrades to existing aircraft means testing their selected systems [4-5] or entire machines on stationary test platforms [6]. In order to carry out aircraft tests, stationary test stands are constructed to simulate actual conditions during selected phases of flight [7]. Simulation tests such as numerical fluid mechanics [8-9] or strength computation [10-11] are an essential element of engineering work, and bench testing is the next stage [12]. The test stand for examination of rotorcraft equipped with active blade control [13-15] requires special and unique constructional solutions which should be verified for their strength.

Flight parameters and efficiency of helicopter hovering can be improved with rotor blades of variable geometry [16-18]. Deformation of the blade during hovering requires actuators embodied in the blade structure [19-21]. The use of actuators means, in turn, the use of an electronic control system for its activation and adaptation of the helicopter head and shaft to mount them. It is mainly connected with placing electric cables in the hub and shaft of the rotor to power these actuators. Helicopters with an electric defrosting system are equipped with a hollow propulsion shaft which connects an angular gear with a hub. Such shafts have inside a fixed shield tube with electric cables to power the system [22]. This tube can also be used to power blade systems. The test stand is not equipped with an angular gear.

The proposed solution means making two technological holes in the shaft wall to lead out the wires, which will impact the stress distribution inside the shaft and its strength. Therefore, a strength analysis of the developed structure had to be performed.
2. Methodology
The strength calculations of the designed elements are performed using the finite element method (FEM), based on the NX NASTRAN linear calculation solver (sol 101). As the numerical calculations are preliminary (comparative), a linear material model and linear approximation of the element's deformation line are used.

2.1. Research object
Figure 1 shows the test object in its basic reference version (without holes). The shaft assembly consists of a shaft (2) and a nut (3) holding the hub on the shaft and a nut holding the rotor shaft in the bearing unit (1).

![Figure 1. Model of the rotor shaft.](image)

The rotor shaft is one of the most overloaded parts of the rotorcraft. It transfers both torque from the engine to the rotor, bending moments and thrust force. The model discretisation, the load distribution and the definition of boundary conditions representing the supporting bearings and rotor head with the blades were performed. Figure 2 shows the geometry prepared for testing. The fixing areas are marked in blue. Points (1) and (2) are the bearing rotation nodes; point (3) is the spline connection between the shaft and the rotor head. Red arrows (4) mark the area of application of torque of 200 Nm, while point (5) marks the nut plane which is affected by an axial force of 1500 N.

The next stage of research included preparing a discrete model, i.e. dividing the model into a finite number of tetra-type elements of an appropriate number to ensure a minimum level of skewness quality at the level of 0.55, which ensured optimal accuracy and speed of calculations. The created grid should reflect appropriate repeatable density for each of the analysed components to obtain correct results. The shaft model is divided into elements with a maximum size of 2 mm. In addition, the grid was densified up to 0.5 mm in the area of the spline and in the section of the shaft where the hole was drilled.

![Figure 2. Shaft geometric model.](image)

3. Results and discussion
3.1. Basic shaft – model 1
The results of the numerical analyses for the reference shaft are shown in Figure 3a. The maximum reduced stress determined according to the Von-Mises rule reaches 42.31 MPa and is concentrated along the spline edge (Figure 3b). The stress values in the area where the hole is supposed to be drilled are maintained at 23.37 MPa (Figure 3c). The axial force stretches the shaft by 0.00715 mm (Figure 4).
3.2. Shaft with two holes of 25 mm diameter – model 2

The simplest solution is to drill a radial hole. Figure 5 shows the geometrical model and dimension of such a hole. Analogously to the basic version, the model for calculation was fixed and loaded with torque and axial force. The geometry was divided into a finite number of elements to reflect the geometry in the form of a calculation grid.

![Shaft model with two holes of 25 mm in diameter.](image)

**Figure 5.** Shaft model with two holes of 25 mm in diameter.

The numerical analysis results for the shaft are shown in Figure 6a. The highest reduced stress determined according to the Von-Mises rule reaches 86.06 MPa and is concentrated along the edge of the hole (Figure 6b). The stresses are distributed radially and concentrated mainly along the edge of the hole (Figure 6c). The shaft stretches along the Z-axis by a maximum value of 0.00746 mm (Figure 7).

![Numerical analysis results, a) total shaft, b) the highest stresses, c) stress concentration area.](image)

**Figure 6.** Numerical analysis results, a) total shaft, b) the highest stresses, c) stress concentration area.

![Extension of the shaft along the Z-axis.](image)

**Figure 7.** Extension of the shaft along the Z-axis.
3.3. Shaft with two longitudinal holes – model 3

Another version of the geometry is a shaft with longitudinal holes along the axis of rotation. The geometrical model and the window dimensions are shown in Figure 8.

![Figure 8. Shaft model with two defined longitudinal holes.](image)

The results of the numerical analyses for the developed shaft concept are presented in Figure 9a. The Von-Mises stresses are mainly concentrated inside the end radius of the hole (Figure 9b) and reach 105.32 MPa (Figure 9c). The elongation of the shaft along the Z-axis is 0.0080 mm (Figure 10).

![Figure 9. Numerical analysis results, a) total shaft, b) the highest stresses, c) stress concentration area.](image)

![Figure 10. Extension of the shaft along the Z-axis.](image)

3.4. Shaft with two quadrilateral holes – model 4

The holes in the next version were in a radial quadrilateral shape with a radial edge end along the shaft axis. The 3D geometrical model with the basic window dimensions is shown in Figure 11.

![Figure 11. Shaft model with two quadrilateral holes.](image)

Figure 12a shows the results of numerical analyses for the developed shaft concept. The zoom of the shaft section with the hole is shown in Figure 12b. The highest stresses reach 69.57 MPa and are concentrated along the inner edge along the lower line forming the wire duct of the spline (Figure 12c). The shaft is extended by 0.00744 mm under load (Figure 13).
3.5. Shaft with four holes of 17.5 mm diameter – model 5
The next solution which meets the structural requirements and can reduce the stresses unlike the previous designs is four holes arranged radially every 90° around the circumference of the shaft. The geometry of the shaft in a configuration of four holes with a diameter of 17.5 mm each is shown in Figure 14. Similarly to the basic version, the model for the calculation was fixed and loaded with torque and lift.

Figure 12. Numerical analysis results, a) total shaft, b) the highest stresses, c) stress concentration area.  
Figure 13. Extension of the shaft along the Z-axis.

Figure 14. Shaft model with four holes of 17.5 mm in diameter.

Figure 15. Numerical analysis results, a) total shaft, b) the highest stresses, c) stress concentration area.  
Figure 16. Extension of the shaft along the Z-axis.
The results of the numerical analyses for the new shaft concept are shown in Figure 15a. Similarly to the previous solutions, the stresses are distributed radially around the hole (Figure 15b). The highest reduced stress, Von-Mises, is concentrated along the edge of the hole and reaches 72.06 MPa (Figure 15c). The shaft stretches along the Z-axis by a maximum value of 0.00743 mm (Figure 16).

3.6. Shaft with two holes of 25 mm in diameter and with the ring – model 6

The culvert hole significantly weakens the shaft structure so if a stress distribution is to be maintained similar to a shaft without holes, it is necessary to strengthen the part weakened by holes. Increasing the shaft diameter in this area is not possible due to the technology of mounting the control disc on the shaft. A clamping ring, e.g. Clampex KTR 700 50x90 (Figure 17) for shaft connection can be used here.

Figure 17. Clamping ring Clampex KTR 700.

The ring and shaft must be drilled in the same way as in the first concept. Figure 18 shows how to mount the ring on the shaft.

Figure 18. Shaft model with two holes and the ring.

The numerical analysis results for the ring shaft are shown in Figure 19a. The stress distribution has changed. The highest stresses are concentrated along the shaft diameter change in the central part of the structure and reach the maximum value of 41 MPa. The next area of the concentrated stresses is the beginning of the spline. The stress here increases to 35.07 MPa (Figure 19b). The application of the ring where the wire holes were drilled reduced the stress to 18.56 MPa (Figure 19c). The shaft elongation along the Z-axis reaches the maximum value equal to 0.00682 mm (Figure 20). Table 1 summarises all obtained analysis results.
Figure 19. Numerical analysis results, a) total shaft, b) the highest stresses, c) stress concentration area.

Figure 20. Extension of the shaft along the Z-axis.

Table 1. Results of FEM analysis.

| No.   | Stress [MPa] | Elongation [mm] | No.   | Stress [MPa] | Elongation [mm] |
|-------|--------------|-----------------|-------|--------------|-----------------|
| Model 1 | 23.37        | 0.00715         | Model 4 | 69.57        | 0.00744         |
| Model 2 | 86.06        | 0.00746         | Model 5 | 72.06        | 0.00743         |
| Model 3 | 105.32       | 0.00800         | Model 6 | 18.56        | 0.00682         |

4. Conclusions

The cable holes drilled in the side wall of the shaft have a significant impact on the stress distribution in the structure and the elongation of the shaft and thus on the load bearing capacity of the construction. The conducted numerical analyses were aimed at reducing the stress resulting from the placement of a notch in the shaft structure within the holes. An additional element such as a clamping ring significantly improved the strength and thus fatigue life of the assembly. The numerical analysis has shown that the stresses in this part of the shaft were reduced by 21% in comparison with the basic structure and the shaft elongation decreased by 4.60% despite the drilled hole.

The analysis has shown that the conceptual solution with an additional stiffening ring is a solution that meets the expectations towards the structure and can be used to construct a test stand for the testing of main rotors with active control blades.

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] Hassanalian M and Abdelkefi A. 2017 Progress in Aerospace Sciences doi:10.1016/j.paerosci.2017.04.003.
[2] Saeed AS, Younes AB, Islam S, Dias J, Seneviratne L and Cai G 2015 Int Conf Unmanned Aircr Syst ICUAS 806–815.
[3] Mare J-C 2019 Chinese J Aeronaut 32 186–199.
[4] Siadkowska K 2020 IEEE 7th MetroAeroSpace doi:10.1109/MetroAeroSpace48742.2020.9160131.
[5] Dopona M, Foxhall N, Dutzler C 2012 SAE Tech Pap doi:10.4271/2012-32-0049.
[6] Czyż Z, Siadkowska K 2020 IEEE 7th MetroAeroSpace doi:10.1109/MetroAeroSpace48742.
2020.9160062.

[7] Czyż Z, Grabowski Ł, Pietrykowski K, Czarnigowski J and Porzak M 2018 Meas J Int Meas Confed 113 46–52.
[8] Czyż Z, Siadkowska K and Sochaczewski R 2019 MATEC Web Conf 252:04002.
[9] Comis Da Ronco C, Ponza R and Benini E 2015 Comput Methods Appl Mech Eng 285 255–290.
[10] Kumar S and Mishra PC 2016 Aerosp Sci Technol 53:252–266.
[11] Skiba K 2019 IOP Conf Ser Mater Sci Eng Pap doi:10.1088/1757-899X/710/1/012003.
[12] Grabowski Ł, Siadkowska K and Skiba K 2019 MATEC Web Conf 252:05007.
[13] Siadkowska K, Raczyński R and Wendeker M 2019 IOP Conf Ser Mater Sci Eng Pap doi:10.1088/1757-899X/710/1/012009.
[14] Wilbur ML and Wilkie WK 2006 Transform Sci Technol Curr Futur Force 185–192.
[15] Fortini A, Suman A, Merlin M and Garagnani GL 2015 Mater Des 85 785–795.
[16] Latalski J 2011 Eksploat i Niezawodn 52 72–78.
[17] Ajaj RM, Beaverstock CS and Friswell MI 2016 Aerosp Sci Technol 49 154–166.
[18] Wilbur ML, Mistry MP, Lorber PF, Blackwell R, Barbarino S, Lawrence TH, Arnold U 2018 Morphing Wing Technol 759–797.
[19] Rafiee M, Nitzsche F and Labrosse M 2017 Thin-Walled Struct 119 795–819.
[20] Tobushi H, Pieczyska E, Miyamoto K and Mitsui K 2013 J Alloys Compd 577 745–748.
[21] Kovalovs A, Barkanov E, Ruchevskis S and Wesolowski M 2017 Procedia Eng 178 85–95.
[22] Kurdelski M and Gębura A 2013 Research Works of Air Force Institute of Technology 33 95-125.