

PARAMETRIZATION OF THE MAIN ROTOR AND WORKING  
ENVIRONMENT FOR DIFFERENT FLIGHT CONDITIONS –  
COMPUTATIONAL FLUID DYNAMICS ANALYSIS AS AN APPLICATION  
FOR MULTIDISCIPLINARY OPTIMIZATION

JAKUB KOCJAN

*Military University of Technology, Doctoral School, Warsaw, Poland*  
*e-mail: jakub.kocjan@wat.edu.pl (corresponding author)*

STANISŁAW KACHEL, ROBERT ROGÓLSKI

*Military University of Technology, Faculty of Mechatronics, Armament and Aerospace, Warsaw, Poland*  
*e-mail: stanislaw.kachel@wat.edu.pl; robert.rogolski@wat.edu.pl*

The paper shows a method of aerodynamic modelling of the whole rotor and exemplary results obtained from complex analyses. The analytical basis of rotor aerodynamics for different phases of the helicopter flight is shown. The analytical calculations are provided to model a single blade motion according to its azimuth angle and to validate the obtained results. The parametric design method is shown to be applied for different blade planform shapes and various section airfoils. The Computational Fluid Dynamics (CFD) fluid domain for the flow around the blade is also prepared using a parametric method. The parametric graphic script is developed to create the flow domain for a one-blade simulation or for a complete  $n$ -bladed rotor effect. The obtained blade model with enclosure is implemented into CFD environment. The method for fluid mesh preparation and the way of defining its properties are given. The simulation is carried out as transient for the  $n$ -bladed rotor. In this simulation, various flight conditions are taken into account. Real rotary motion of the blades is simulated with artificially enforced mesh motion. The obtained numerical results are compared then with analytical assumptions. The simulation findings which are the inputs for further analysis are shown with graphical representations. As an output of the research, new options for main rotor optimization are developed. The usage of combined parametric modelling confirmed with aerodynamic analysis for different flight conditions is shown in the work as a new perspective for design optimization of the main rotor.

*Keywords:* helicopter, main rotor, rotor blade, geometric modelling, computational fluid dynamics (CFD)

## 1. Introduction

The Russian invasion on Ukraine revised the usage of helicopters on modern battlefield. The need of preparing constructions that are capable to provide better features for comprehensive military operations is observed more than in any other conflict in XXI's century. As a consequence, a lot of countries are improving existing structures and searching for brand new solutions of VTOL vehicles (Oh *et al.*, 2021). Therefore, also in Polish Military University of Technology, a development of rotorcraft design methods is being conducted as an activity included in a complex research program which is aimed at finding new approaches and design solutions for needs of structural or aeroelastic optimization. This work is a second part of the recalled research program. The first one was focused on preparing a parametric blade model and its analytical and CFD validation for further analysis. The research were preceded with comparative

analysis of commonly used helicopter configurations. The results of evaluation and proposals for construction improvements were published in (Kachel *et al.*, 2021).

The parametrization of the blade provides a less time-consuming solution for the preliminary design phase. The advantages of parametric blade modeling were demonstrated in numerous publications including (Sagimbayev *et al.*, 2021; Ma *et al.*, 2021). What is more in this stage of the study, the fluid enclosures for CFD analysis are also parametrized and generated by the prepared program. The usage of parametric models to fulfill the required features of shape and structure is shown in (Lim, 2018; Allen *et al.*, 2021) for airfoil design, (Tixadou, 2021; Grebenikov *et al.*, 2021) for strength design and (Bailly *et al.*, 2019) for aerodynamics. Some model tasks from the above works defined with parametrization declared both for aerodynamic shape, blade structure and fluid domain are highly advanced numerical problems. Solving such problems generally demands applying Fluid Structure Interactions (FSI) techniques, which gives optional results varied due to different design configurations. Simulations of the rotor aerodynamic and aeroelastic effects, taking into account sensitivity analysis due to key design parameters, significantly support the decision-making process in relation to the selection of optimal utility variants.

In the research described here, for preparation parametric models and fluid enclosures, some specific programming language was used. Examples of parametrization in the modelling using GRIP language were shown in (Ryazanov, 2016; Shabliy and Dmitrieva, 2014; Grabowik *et al.*, 2015). Open GRIP (Graphic Interactive Programming) available in Siemens NX CAD environment is a language that provides a possibility to create an external solid body or internal structure geometry of any airframe part. Virtual elements are generated with the use of commands and parameters defined with a specific programming code. The GRIP language is dedicated for research processes, because it grants an option for inertia properties of the generated model. The obtained model features can be used for aerodynamic or strength analyses carried out in further stages of the simulations. The logical conditions that are enclosed in the program code can be used to prepare loops which will provide a possibility of strength calculation with fitting the best dimensions. As an output, a geometry generator is prepared with initial optimization functions. Other main rotor optimization studies with focusing on aerodynamic optimization were presented in (Stalewski, 2017a,b; Xie *et al.*, 2017; Stalewski and Zalewski, 2019; Okumus *et al.*, 2022) while optimization procedure for best flight performance was shown in (Slavik *et al.*, 2029).

The aim of the research presented herein is to analyze the aerodynamic rotor model in different flight conditions. The new approach is to provide geometry with enclosures prepared to be applied into CFD environment, where different flight conditions with real main rotor motion could be evaluated and compared. One of the examples of CFD usage for strength analysis was presented in (Spyropoulos *et al.*, 2021). There are some main research institutes that are focused on main rotor optimization problems, ONERA French Aerospace Lab and DLR German Aerospace Center for instance. They published several methods of solving the optimization problem using CFD methods (Goerke *et al.*, 2012; Wilke, 2021; Jain, 2022). However, there were no parametrization of the rotor blade shape, position and entire fluid domains simultaneously for given flight conditions, which is the main goal of this research in order to provide a comprehensive tool for future combined optimization solutions.

As it was mentioned above, the solutions for the optimization process which are being developed in the mentioned research program, started with evaluating the blade and assessing the methods for further analysis. The preliminary results from the initial stage of investigations were published in (Kocjan *et al.*, 2022). In this research, geometry of the existing rotor was taken as an example for evaluating the proposed solution. The GRIP program code for the modeling was developed by adding the option for enclosures preparation and expand possibilities to enter more parameters of the main rotor by end-user. The dimensions of enclosures are adequate to rotor dimensions.

The prepared model will be a basis for fluid dynamics computation using Ansys Fluent. The parameters of main rotor movement will be calculated from analytic formulae. The value of collective pitch control and a change of cyclic control are calculated with the use of a MATLAB code. Then the calculated values are inputs for CAD model generation and then for CFD motion simulation. Furtherly, the mesh for numerical simulation is generated with the Fluent meshing tool. The mesh size decreases as it approaches the blade surface. The inflation model is also applied. In the research, a poly-hexcore mesh was used. The prepared mesh is configured for calculation using sliding mesh options. The mesh motion is described with outcomes of analytical calculations, which were earlier prepared. During the studies, the mesh size was adapted to obtain best results within a reasonable time. The results were compared with the assumptions. The lift and drag values were used to evaluate the model and to check if the parametrized rotor model with air domains is applicable to CFD environment and if the obtained numerical results, especially pressure distributions, are reasonable and eventually acceptable. To the best of co-authors' knowledge, similar research merging the proposed methods for the helicopter main rotor design and main rotor optimization operation has not been performed so far.

This paper is organized in a specifically intended way. The research methods are described in Sections 2 and 3, where the mathematical model, which is the basis for parametric programming, is defined. In Section 3, the parametric programming is described. The mesh properties and simulation setup are presented in Section 4. The results and evaluation of the method is discussed in Section 5. The research methods and outputs, with application to next phases of the program are concluded in Section 6.

## 2. Analytical model for parametrization

### 2.1. Mathematical model

The conditions of operation of the main rotor are complex and impose on a designer a necessity to check all of the working aspects. Before performing, an optimization loop for parametrization of main rotor features must be introduced.

The first data to be introduced are blade parameters. To obtain required operational capabilities, the blades construction requires to take into consideration both geometric and aerodynamic characteristics. The flow periodic changes, flexibility of blades and conditions of operation in the presence of a strong centrifugal force imply a relationship between pressure loads and deformations of the material. The blades were well parametrized in the previous step of the research (Kocjan *et al.*, 2022), the deformations are going to be considered in the further phases.

In the presented studies, the conditions of the  $n$ -bladed main rotor operation are analyzed and transformed as input parameters for geometry generation and numerical CFD simulation. To prepare a rotor model which is ready to be implemented into Ansys Fluent, in accordance with (Johnson, 1994; Bramwell *et al.*, 2001), pitch angles should be calculated for the main rotor trim conditions. The trim of the rotor is calculated from the thrust, pitching moment for lateral and longitudinal movement

$$\theta(\psi) = \theta_0 - A_1 \cos \psi - B_1 \sin \psi \quad (2.1)$$

where the angles are calculated for different flight conditions. The helicopter speed is given in calculation as a dimensionless speed ratio. To find the longitudinal cycling pitch angle, the first step is to calculate the  $H$ -force coefficient in the disc plane

$$h_{cD} = \frac{1}{4} \mu \delta \quad (2.2)$$

As a result, the disc incidence can be calculated as follows

$$\alpha_D = -\frac{1}{w_c} \left( \frac{1}{2} \mu^2 d_0 + h_{cD} \right) \quad (2.3)$$

where  $d_0$  is the fuselage drag ratio and  $w_c$  is the weight coefficient. Now the mean inflow ratio relative to the disc plane can be obtained in the following way

$$\lambda_D = (\mu\alpha)_D - \lambda_i \quad (2.4)$$

The next step for the rotor trim is to calculate the collective pitch angle which will be used for GRIP program to prepare the main rotor geometry installed for designated flight conditions. The collective pitch is calculated from the main rotor thrust expression

$$t_{cd} = \frac{a}{4} \left( \frac{2}{3} \theta_0 \frac{1 - \mu^2 + \frac{9\mu^4}{4}}{1 + \frac{3\mu^2}{2}} + \lambda_D \frac{1 - \frac{\mu^2}{2}}{1 + \frac{3\mu^2}{2}} \right) \quad (2.5)$$

Further, in order to obtain the longitudinal cycling pitch angle, it is crucial to calculate the flapping coefficient

$$a_1 = \frac{2\mu \left( \frac{4\theta_0}{3} + \lambda_D \right)}{1 + \frac{3\mu^2}{2}} \quad (2.6)$$

Therefore, the  $B_1$  angle (with the assumptions for a change of the center of gravity –  $h$  and  $f$ ) is ready to be calculated

$$B_1 = \frac{a_1 + C_{mf} + h_{cD}h - w_c f}{w_c + C_{ms}} \quad (2.7)$$

where  $C_{mf}$  is the pitching moment coefficient for fuselage and  $C_{ms}$  is the pitching moment coefficient due to hinge offset. Then, to get the lateral cyclic pitch angle, the lateral flapping coefficient should be given with the following expression

$$b_1 = \frac{\frac{4}{3}(\mu a_0 + 1.1v^2\lambda_i)}{1 + \frac{\mu^2}{2}} \quad (2.8)$$

In consequence, the lateral cyclic pitch angle can be written as

$$A_1 = -b_1 - \frac{w_c f + \frac{T_t}{W} t_c h_t}{t_c h + C_{ms}} \quad (2.9)$$

At the end of analytical consideration, for the main rotor evaluation, its drag force is needed and it can be calculated from the expression

$$D = \frac{P}{V} + X \quad (2.10)$$

where  $P$  is the power required for the evaluated flight conditions,  $X$  is a force acting in the longitudinal direction. The  $X$ -force can be estimated with the formula

$$X = -T \sin \alpha_D - H_D \cos \alpha_D$$

## 2.2. Analytical calculations for preparing the simulation

All calculation formulae described above were all prepared in MATLAB software. The outputs of calculations are a basis for geometry generation in GRIP program and then for model configurations applicable in Ansys Fluent.

It starts with the input parameters of the designed blade and flight conditions. The blade planform is defined with a specific polynomial function. Next, the rotor solidity factor is calculated as an integral. Using MATLAB functions required for the collective pitch are possible to be established. The weight coefficient, fuselage drag ratio, inflow velocity and disc incidence are computed. After that, the inflow ratio relative to the disc plane can be processed. The program in the further step is solving equation (2.5) to obtain the collective pitch value. Next stage of the calculations is to estimate the flapping coefficients. The pitching moment coefficient is assessed at the subsequent step. With the flapping and pitching moment coefficient, the  $B_1$  angle is ready to be computed. After obtaining the first angle, the second is calculated similar to the first using the equations mentioned above. The MATLAB source code used for all described calculations is shown in Fig. 1.

The calculated collective pitch is modelled in CAD geometry. The collective pitch angles are a basis for the rotor cyclic pitch movement modelled then in Ansys Fluent.

```

94  a1=deg(rad2deg(a1))
95
96  hcd2=1/4*u*del+a*lamd/4*(1/2*a1-u*tet0)
97
98  alfd2=-((0.5*u^2*d0*cos(tc)+hcd2)/wc-tc)
99  ud2=u*cos(alfd2)
100 lamd2=ud2*alfd2-lami
101
102  a0=y/8*(tet0*(1-19*u^2/18+3*u^4/2)/(1+3*u^2/2)+4/3*lamd2*(1-u^2/2)/(1+3*u^2/2))
103  dtf=(1-sin(alfd2)/(1+sin(alfd2)))
104  b1=4*((u*a0+1.1*dtf^(1/2)*lami)/3)/(1+u^2/2)
105  qc=del*(1+3*u^2)/8-lamd2*wc-u*hcd
106
107  Cms=b*Mb*xg*e/(2*Solid*A*R)
108  B1=a1+(Cmf+hcd2*h-wc*f)/(wc*h+Cms)
109  B1=simplify(B1)
110  axis=simplify(a1-B1)
111  B1=rad2deg(B1(1,1))
112  B1=simplify(B1)
113
114
115  H=hcd2*ro*A*om^2*R^2*Solid
116  P=qc*ro*A*om^3*R^3*Solid
117
118  X=-W*sin(alfd2)-H*cos(alfd2)
119  D=P\|V+X
120  D=simplify(D)
121
122  alfd2=simplify(alfd2)
123  a1=simplify(a1)
124  axis=simplify(rad2deg(a1)-B1)
125
126  alfHT=simplify(D/(W+H/W))
127  alfHT=simplify(rad2deg(alfHT))
128
129
130  Q=qc*ro*Solid*A*om^2*R^3
131
132
133
134  %%mass=@(x) Mb.*(x/(0.79*R)-0.21/0.79)      % mass function
135  %%Inert=@(x) mass(x).*x.^2
136

```

Fig. 1. Example of MATLAB numerical code for preliminary calculations

## 3. Parametric model for simulation process

The main goal of this research is to obtain a new approach for main rotor blade modelling, which will be implemented into the main rotor optimization loop. The goal is achieved using Open

GRIP code, which is implemented into Siemens NX. The program was taken from the first stage of the studies and improved. The program algorithm is shown in Fig. 2.

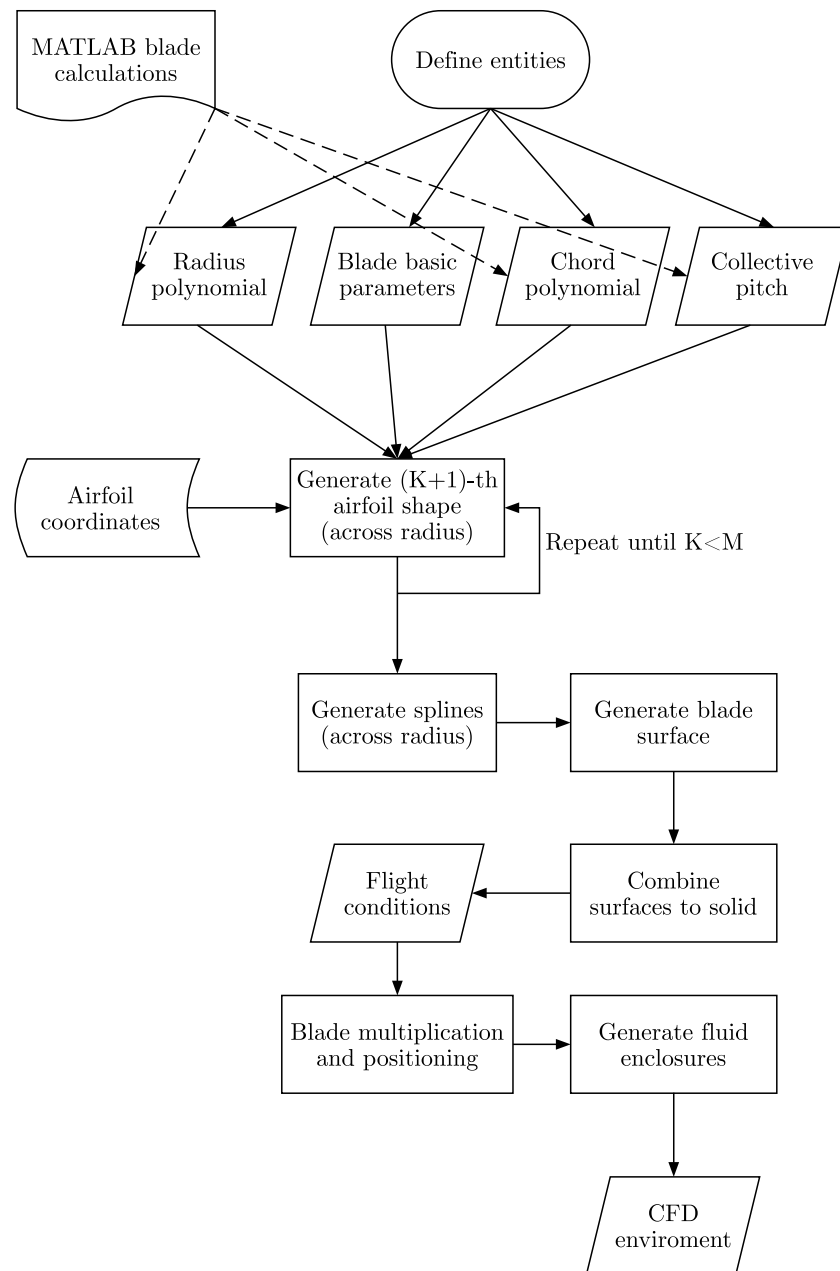


Fig. 2. Enhanced algorithm for generating parametric geometry model

The geometry of the model is parametrized in the program using all main rotor geometrical dimensions. The user is able to enter the blade basic parameters into a popup input window. The first parameters are the number of blades, rotor radius and blade chord. The airfoil is generated from the text file with coordinates, so the user can prepare the blade for any chosen airfoil geometry. The next stage of parametrization is to enter the polynomial coefficient for blade twist and blade chord, so the blade chord and twist can change along the span. With the prepared solution, the blade shape can be freely changed with accordance to analytically calculated values. The windows with applets for introducing control parameters are shown in Figure 5. The shape generation is based on inbuilt GRIP commands for the line and splines,

which provides the desired shape. The number of sections to create the geometry can also be chosen. An example of generating the blade geometry is shown in Fig. 3.

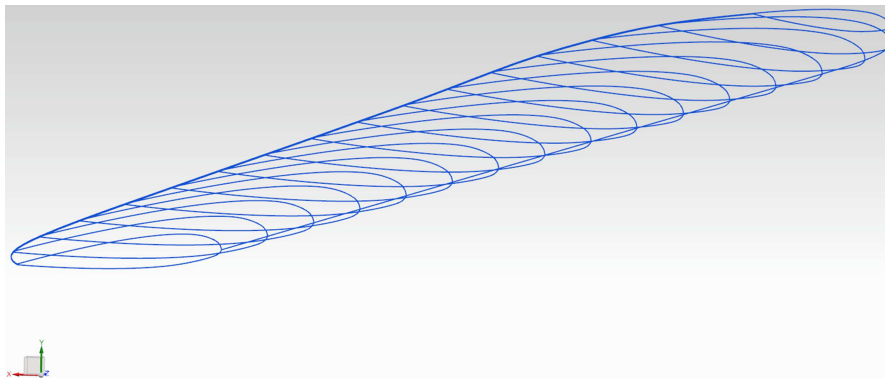


Fig. 3. Blade parametrisation

The second stage of parametrization is the  $n$ -bladed rotor. The blades are positioned with the angles that were trimmed analytically. Each blade is positioned at a different angle in accordance to their initial azimuthal position in simulation.

The next phase is generation of fluid enclosures for multi-moving mesh simulation. In the code, the volumes of fluid domains are programmed proportionally to the rotor dimensions. The domains are generated as follows: the main air enclosure as a cuboid for forward flight and as a cylinder for hover, rotor enclosure as a cylinder with each blade having its own cylinder domain. The space geometry of applicable enclosure variants is presented in Fig. 4. The selectable flight conditions – forward flight or hovering are possible to be chosen by the user.

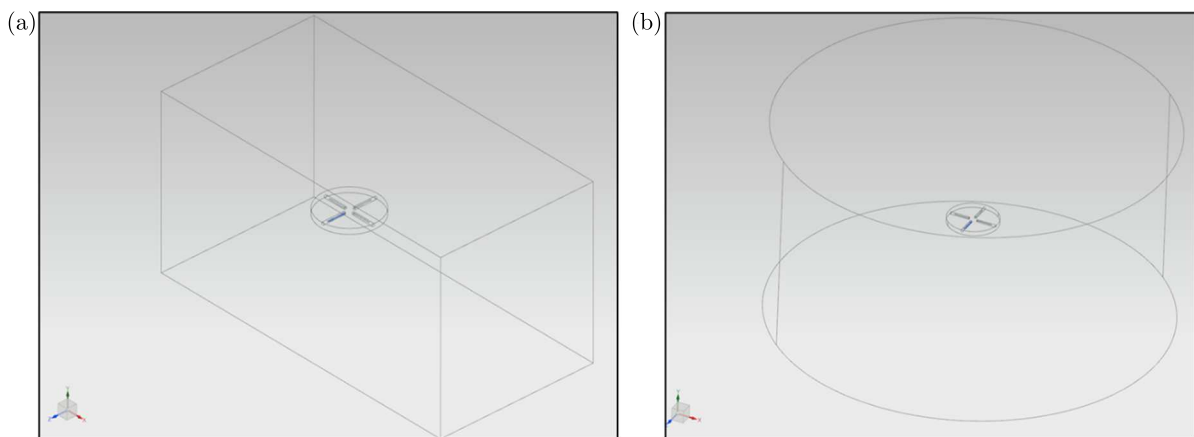


Fig. 4. Fluid domains: (a) for forward flight, (b) for hovering flight

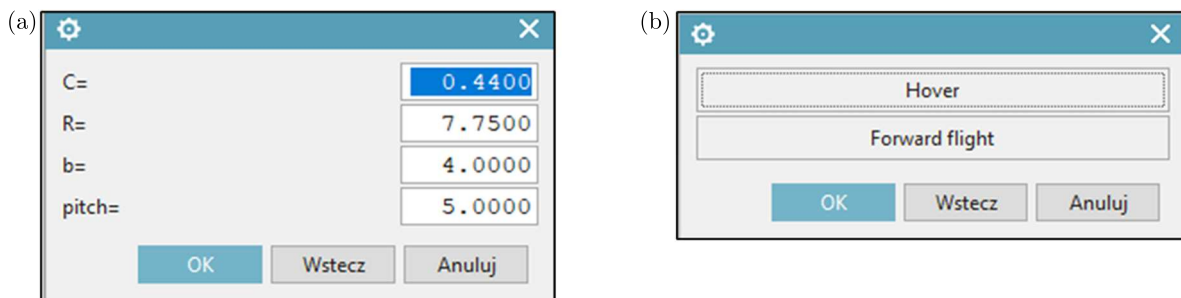


Fig. 5. User input options: (a) blade parameters, (b) flight conditions

The ready to simulation geometry is generated in few seconds, so the designer can prepare different models for CFD analysis in a short time.

#### 4. Mesh and simulation properties

The model for CFD simulations was prepared in Ansys Fluent. The generated geometry with fluid domains was imported into the mentioned environment. To generate the fluid mesh of finite volume elements, the Fluent Mesher was used. It was chosen because of the future automatization of the process. This software provides a full path of activities that are necessary to generate the ready-to-use mesh. Body sizing, face sizing and inflation options were used to prepare a correct simulation. The boundary layer was generated using the aspect ratio options. The air enclosure size was calculated from the rotor dimensions. The ratios chosen to obtain correct domains were defined in the following way:  $X$  for the air enclosure,  $Y$  for the rotor enclosure and  $Z$  for the blade enclosures. The maximum size of a single volume element was set as 5000 mm, the elements on the blade were set to a value of 10 cm spanwise and chordwise. The inflation was set for 15 layers with 80 aspect ratio. The  $y+$  value was established at an average value of 300, which gave the boundary layer a wall function. The mesh was generated as a poly-hexcore mesh and consisted of approximately  $10^6$  elements (depending on the model). An example of the prepared mesh is shown in Fig. 6.



Fig. 6. Polyhexcore mesh of fluid volume elements

The domains were named adequate to their destinations. The velocity inlet for the forward flight was set as a pressure far-field – velocity inlet, and the end of the cuboid was provided as a pressure outlet. The pressure far-field – velocity inlet option in Ansys Fluent provides a possibility to set the inlet velocity direction and magnitude with the vector coordinates. The contact regions were named to identify the internal connection between the domains in the preparing of the solution.  $K-\omega$  SST viscosity model was chosen for the simulation, because it is recommended for solving CFD rotor tasks. The viscosity model results with the  $y+$  for wall function boundary layer treatment were checked whether it gave satisfying results without extending calculation, what would be for more complex boundary layer simulations. The time step for transient simulation was set at 0.001 s. The convergence for each time step was settled for continuity at  $10^{-5}$  with 100 iterations. The fluid domain with declared boundary conditions and the rotor model inside is presented in Fig. 7.



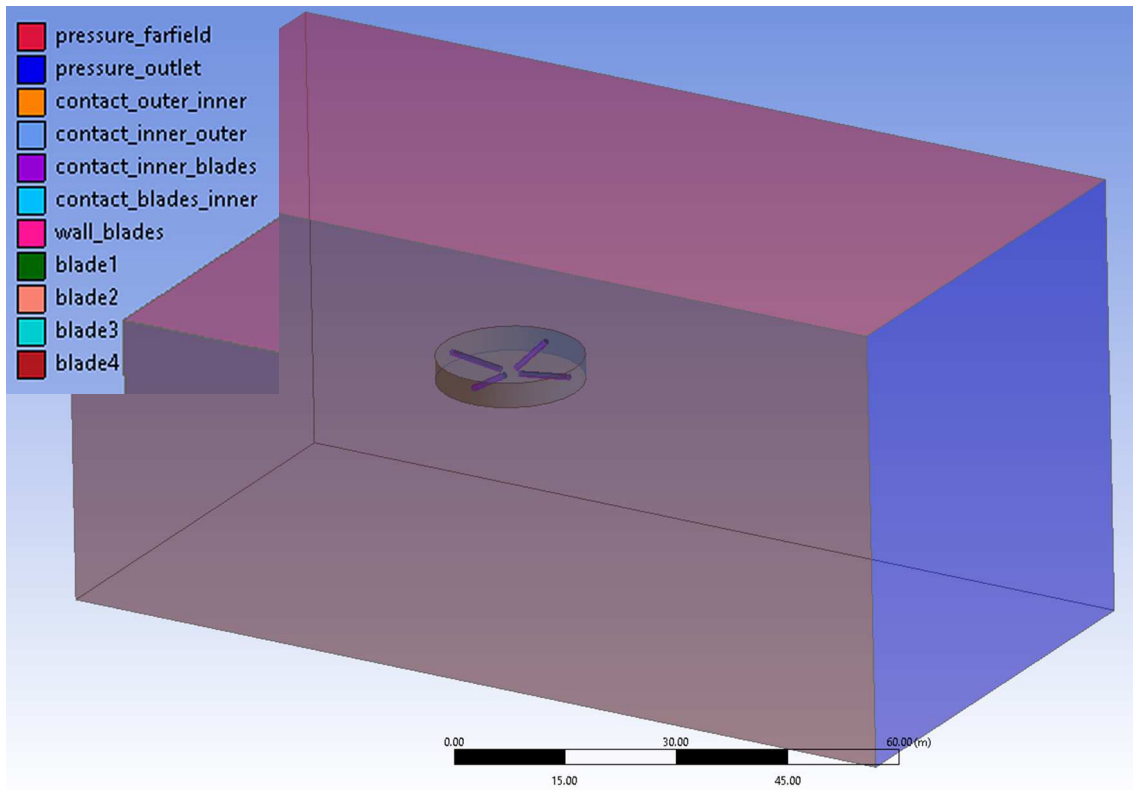


Fig. 7. The fluid domain with boundary conditions and the rotor model enclosure defined for the CFD process

## 5. Results

As a basis for CFD calculations, the rotor geometry of the rotorcraft W-3 Sokol was taken. The parameters of the helicopter are: radius – 7.85 m, chord – 0.44 m, rotational speed – 255 rpm, weight 6400 kg. The lift force should be adequate to the aircraft weight and the drag force is calculated from analytical assumptions. The first cut calculation was made for the existing rotor dimensions and features. The rotor position and movement were calculated in MATLAB program described in Section 2.2. For evaluation of the method, various rotor dimensions and flight parameters were used. The angles for the preparation of simulation are shown in Table 1.

**Table 1.** Calculated angles for simulation

Case No.	Collective [deg]	$B_1$ [deg]	$A_1$ [deg]	Tip path angle [deg]
1	11.12	3.46	-3.72	10.75
2	11.52	5.26	-4.87	10.71
3	8.31	2.52	-3.45	7.44
4	9.85	5.48	-5.48	7.43
5	9.48	3.10	-4.38	7.83

The calculated results are presented in Table 2. It shows that the assumptions which were made for the movement simulation and calculations are correct. The values of basic rotor forces are similar to those calculated, the average difference between the results and analytical calculations is at 4%. The comparison is made with analytical calculations of the drag force and

helicopter weight. The lift force that is required for flight of the helicopter is generated adequately to the needs.

**Table 2.** Simulation results

Case No.	Analytical assumptions					CFD simulation	
	Weight	MR radius [m]	Blade chord [m]	Flight speed [km/h]	Drag [N]	Lift [N]	Drag [N]
1	62784	7.85	0.3	100	1276	61706	1256
2	62784	7.85	0.3	150	2870	63708.25	2982
3	62784	7.85	0.44	100	1276	61537	1139
4	62784	7.85	0.44	200	5098	62625	5132
5	62784	7	0.6	100	1276	66850	1295

In consequence, working conditions of rotor blades can be evaluated using the obtained results. The major finding, that will be used in optimization loop (which will be aimed at mass reduction of the blades and stiffness improvement), is the pressure distribution. Examples of pressure distributions are shown in Fig. 8.

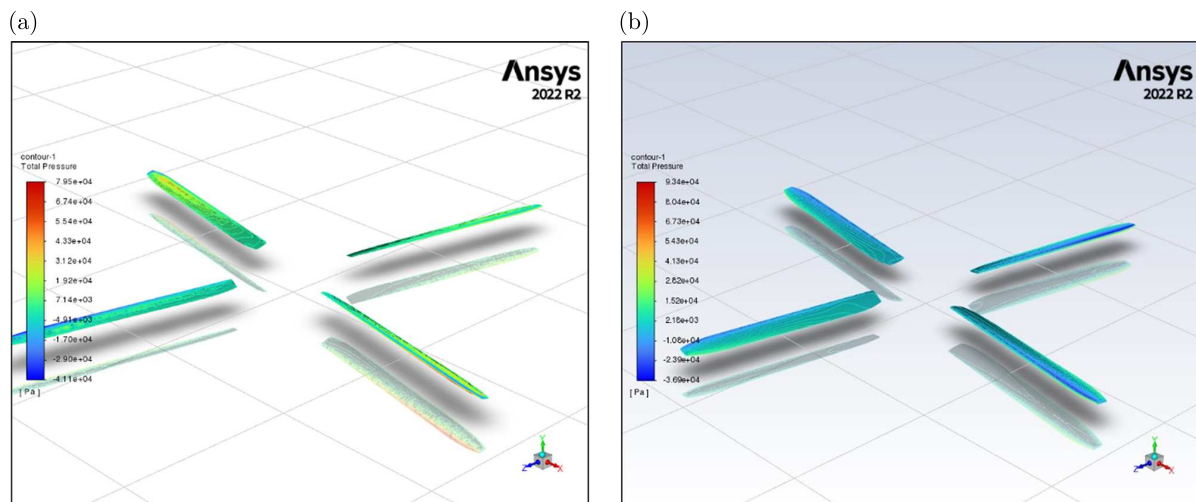


Fig. 8. Pressure distribution: (a) case 1, (b) case 2

The pressure distribution for the calculated flight case enables determination of the load acting on rotor blade surfaces during their rotary movement. Different flight conditions and different blade geometry can be evaluated by the designer. To obtain these results with application of the proposed method, 6 to 8 hours is needed, where 15 minutes is 3D model preparation, 45 min to 2 h is mesh and simulation preparation, the rest is Fluent processing time (which is also depending on mesh parameters).

The proposed way of model preparation and calculation of simulation parameters provides information about the influence of parametrisation on the obtained dimensions. The required drag and lift are the same for different flight parameters, however the angles of rotor disc and blade pitch positions are changing depending on blades dimensions. With CFD simulation, the proposed modelling was proved to be correct and applicable to next steps of rotor optimisation.

## 6. Conclusions

As it is described at the beginning of this paper, this research is a part of the program aimed at finding new solutions for rotorcraft structures design using optimization methods. This is a

next step for preparing a full rotor model which will be optimized to obtain the best mass and stiffness characteristics.

It is confirmed that developing the rotor model with fluid enclosures, using parametric modelling, provides a versatile tool to quickly prepare different rotor geometries for CFD analysis. The code properties provide a possibility to prepare correct fluid domains. The enclosures are generated proportionally to rotor dimensions. The Open GRIP syntax Boolean options, to cut out the volumes from wider areas, enables an exact model preparation, which is ready to be analyzed using dedicated software. The model compatibility problem with software did not occur during the calculations. Using the parametric modeling significantly reduced the time of simulation preparation for different geometries and flight conditions.

The CFD analysis confirmed the expected results. The mathematical model of the rotor movement led to similar forces that were inputs for analytical calculations. The generated domains and models were easily configured for mesh simulation. The interior contacts for mesh interaction during simulation of motion were correctly corresponding with the indented contact regions.

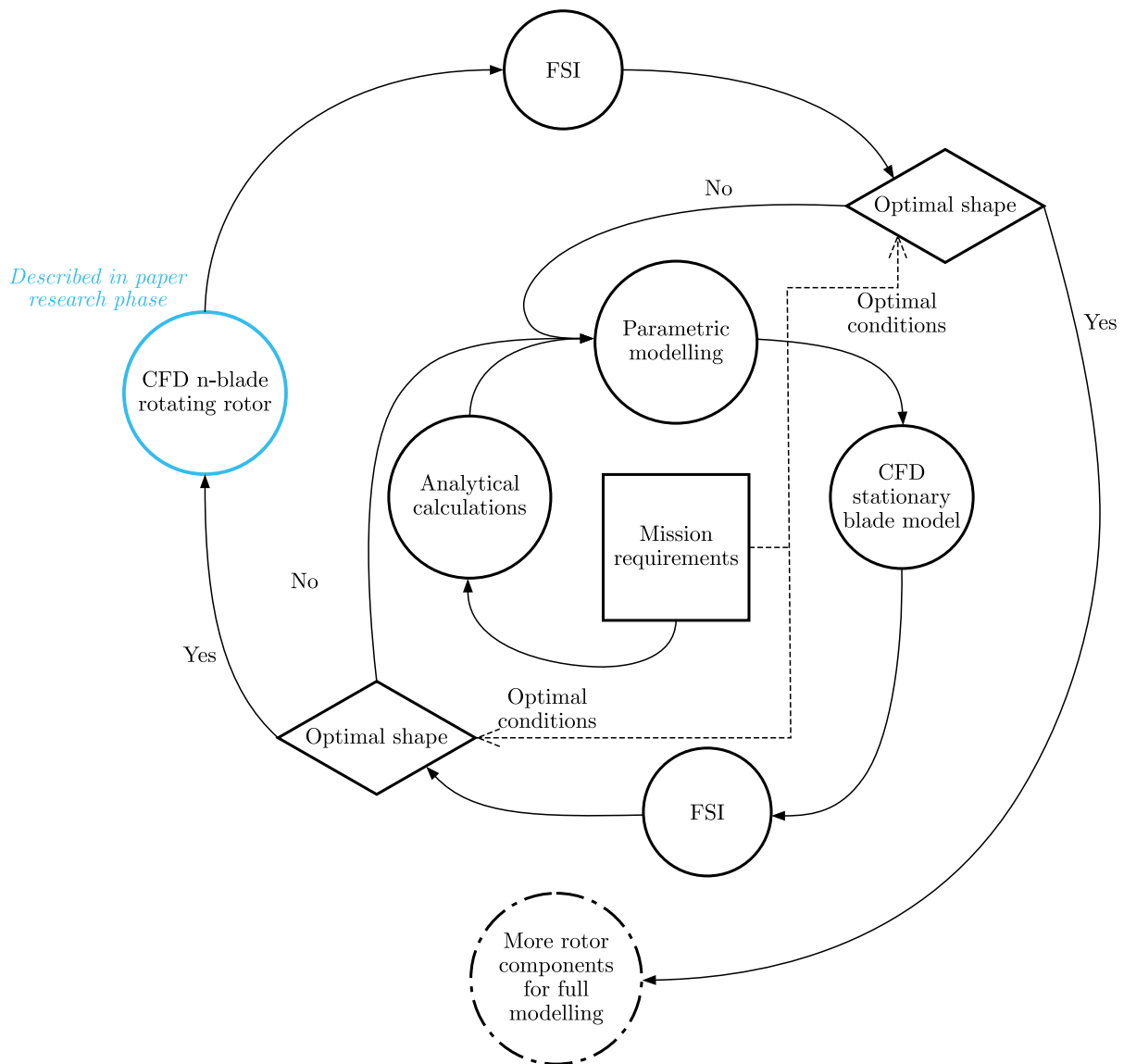


Fig. 9. Design spiral for rotor optimization

The simulation results showed that the model was correctly prepared. However, providing all required conditions to replicate the working environment of the real rotor, the simulation of the rotor movement and a change of pitch angles was time and memory consuming. But in fact it is a necessary condition to obtain correct results.

The studies shows a new way of preparing and conducting main rotor CFD simulations using a parametric approach. The parametric modelling reduces the total time of simulations. The results obtained showed the working conditions of the main rotor blades in terms of pressures, velocity or turbulence. The outputs can now be combined in Ansys with FEM analysis and calculated simultaneously using the FSI procedure.

The advantage of the procedure is fidelity of the simulation. This kind of simulations provides high fidelity results, that can be used to evaluate parameters of prepared blades such us the shape and strength structure. This will be a next phase of the proposed optimization loop. The application of the studies described in this paper is only a one of many activities included in the optimization procedure illustrated with the so called design spiral. It is shown in Fig. 9.

#### *Acknowledgement*

The methods and results presented in this paper have been obtained during research works conducted within a university research project entitled “Methods of optimal design of aircraft to improve its structural and aerodynamic properties”. It was financed by the Military University of Technology (Warsaw, Poland), in in 2022 and 2023, under the university research project UGB-782/2022 and UGB-819/2023.

#### *Data Availability Statement*

Program code is available at: <https://github.com/jakubkocjan/parametricrotordesign> (accessed on 30 November 2022).

The work and its results were presented during the conference “Mechanika w Lotnictwie, ML-XX 2022”.

## References

1. ALLEN L.D., LIM J.W., HAEHNEL R.B., DETTWILLER I.D., 2021, *Rotor Blade Design Framework for Airfoil Shape Optimization with Performance Considerations*, Engineer Research and Development Center
2. BAILLY J., BAILLY D., DIDIER B., 2019, Multifidelity aerodynamic optimization of a helicopter rotor blade, *AIAA Journal*, **57**, 8, 3132-3144
3. BRAMWELL A.R.S., DONE G., DAVID B., 2011, *Bramwell's Helicopter Dynamic*, Butterworth-Heinemann, Linacre House, Jordan Hill, Oxford
4. GRABOWIK C., KALINOWSKI K., KEMPA W., PAPROCKA I., 2015, A method of computer aided design with self-generative models in NX Siemens environment, *IOP Conference Series: Materials Science and Engineering*, **95**, 1
5. GREBRNIKOV A.G., *et al.*, 2021, Analysis of design and technological features of the main rotor blades of heavy transport helicopters, *Open Information and Computer Integrated Technologies*, **93**
6. GOERKE D., LE DENMAT A.L., SCHMIDT T., KOCIAN F., NICKE S., 2012, Aerodynamic and mechanical optimization of CF/PEEK blades of a counter rotating fan, *Proceedings of the ASME Turbo Expo*, **7**, Parts A and B
7. JAIN R., 2022, Computational fluid dynamics transition models validation for rotors in unsteady flow conditions, *Journal of Aircraft*, **59**, 4
8. JOHNSON W., 1994, *Helicopter Theory*, Dover Publications, New York
9. KACHEL S., ROGÓLSKI R., KOCJAN J., 2021, Review of modern helicopter constructions and an outline of rotorcraft design parameters, *Problems of Mechatronics Armament Aviation Safety Engineering*, **12**, 3

10. KOCJAN J., KACHEL S., ROGÓLSKI R., 2022, Helicopter main rotor blade parametric design for a preliminary aerodynamic analysis supported by CFD or panel method, *Materials*, **15**, 12
11. LIM J.W., 2018, Application of parametric airfoil design for rotor performance improvement, *44th European Rotorcraft Forum 2018, ERF 2018*, **1**
12. MA L., ZHAO Q., ZHANG K., ZHANG X., ZHAO M., 2021, Aeroelastic analysis and structural parametric design of composite rotor blade, *Chinese Journal of Aeronautics*, **34**, 1
13. OH S., PARK D., JI H.M., 2021, Development of preliminary conceptual design/comprehensive analysis programs for next generation rotorcraft, *Journal of the Korean Society for Aeronautical and Space Sciences*, **49**, 1
14. OKUMUŞ O., SENIPEK M., EZERTAS A., 2022, Multi-objective multi-fidelity aerodynamic optimization of helicopter rotor, *AIAA SCITECH 2022 Forum*
15. RYAZANOV A.I., 2016, Automated 3D modeling of working turbine blades, *Russian Engineering Research*, **36**, 9
16. SAGIMBAYEV S., KYLYSHBEK Y., BATAY S., ZHAO Y., FOK S., LEE T.S., 2021, 3D multidisciplinary automated design optimization toolbox for wind turbine blades, *Processes*, **9**, 4
17. SHABLIY L.S., DMITRIEVA I.B., 2014, Conversion of the blade geometrical data from points cloud to the parametric format for optimization problems, *ARPJN Journal of Engineering and Applied Sciences*, **9**, 10
18. SLAVIK S., KLESA J., BRABEC J., 2020, Propeller selection by means of pareto-optimal sets applied to flight performance, *Aerospace*, **7**, 3
19. SPYROPOULOS N., PAPADAKIS G., PROSPATHOPOULOS J.M., RIZIOTIS V.A., 2021, Investigating the level of fidelity of an actuator line model in predicting loads and deflections of rotating blades under uniform free-stream flow, *Applied Sciences*, **11**, 24
20. STALEWSKI W., 2017a, Design and optimisation of main rotor for ultralight helicopter, *Journal of KONES Powertrain and Transport*, **24**, 4
21. STALEWSKI W., 2017b, *Design and aerodynamic optimization of rotorcraft* (in Polish), Warsaw
22. STALEWSKI W., ZALEWSKI W., 2019, Aerodynamic design and optimisation of main rotors for light rotorcrafts, *45th European Rotorcraft Forum 2019, ERF 2019*, **1**
23. TIXADOU E., 2021, Optimisation of the structure of A helicopter blade in view of reducing cabin vibration, *Proceedings of the 47th European Rotorcraft Forum*, Google Scholar
24. WILKE G., 2021, Quieter and Greener rotorcraft: concurrent aerodynamic and acoustic optimization, *CEAS Aeronautical Journal*, **12**, 3
25. XIE J., XIE Z., ZHOU M., QIU J., 2017, Multidisciplinary aerodynamic design of a rotor blade for an optimum rotor speed helicopter, *Applied Sciences*, **7**, 6