This paper is available online at www.praiseworthyprize.org
International Review of Aerospace Engineering (I.RE.AS.E), Vol. 16, N. 4
ISSN 1973-7459 August 2023

Preliminary Numerical Study of a Rectilinear Blade Cascade Flow for a Determination of Aerodynamic Characteristics

M. Majcher, M. Frant, R. Kieszek

Abstract – *In the design process of blade arrangements in axial flow machines, it is extremely important to have the aerodynamic characteristics of the airfoils in a rectilinear blade cascade arrangement. However, the source of these characteristics is often difficult to access. This paper presents a methodology for the numerical determination of the characteristics of the lift coefficient, drag coefficient and flow turning angle of the airflow as a function of the angle of attack, by using the example of a blade cascade formed by NACA 65-010 airfoils for the flow intake angle β_1=30° and the cascade solidity σ=1. A numerical analysis of the impact of the numerical mesh parameters and the applied turbulence model on the obtained values of the lift coefficient, drag coefficient and flow turning angle of the airfoil in a rectilinear blade cascade has been performed. The numerical values obtained have been compared to experimental results. In this respect, satisfactory agreement has been obtained between experimental and numerical results, confirming the feasibility of numerical determination of the aerodynamic characteristics of airfoils in a rectilinear blade cascade.*

Keywords*:* *Computational Fluid Dynamics, Airfoil, Blade Cascade, Aerodynamics Characteristics, Axial Flow Machines*

Nomenclature

b	Cascade width
c	Axial component of velocity
CAD	Computer Aided Design
C_d	Drag coefficient
C_l	Lift coefficient
l	Airfoil chord
t	Tangential spacing between blades
u	Tangential velocity
w	Relative velocity
Δc_u	Tangential velocity increase
α	Angle of attack
β	Angle between flow direction and a perpendicular to the cascade axis
γ	Airfoil angle in the cascade
θ	Flow turning angle ($\beta_2 - \beta_1$)
σ	Cascade solidity (l/t)
Subscripts	
1	Upstream of blade row
2	Downstream of blade row

I. Introduction

Axial flow machines are fluid-flow machines in which there is a continuous flow of the working medium (liquid or gas), and energy is transferred from or to the fluid medium. In addition, these types of working machines are characterised by the fact that the direction of flow of the working medium coincides with their axis of rotation.

Examples of this type of machine are blowers, fans, compressors, and turbines. These machines vary with their pressure rise values and the direction of energy transfer. In the case of compressors, the energy transfer is from the rotor to the fluid and in the case of turbines, it is vice versa: from the fluid to the rotor. A key aspect of the correct operation of axial flow machines is the correct design of rotor and stator rings. This applies in particular to the blade arrangement, whose final configuration results from a correctly executed design process for the given operating parameters. These parameters may include total pressure build-up, volumetric or mass flow rate, generated noise level, operating point efficiency, etc. Properly designed fluid-flow machines will also have a high degree of efficiency, which can translate into reduced energy consumption, for example, in the case of electrically powered fans. The design process of axial flow machines consists primarily of determining the correct geometry of the blade rings in terms of the performance to be achieved by the designed machine. An excellent method that allows the preliminary design of axial flow machines is the analytical method based on a flat rectilinear blade cascade. The rectilinear blade cascade model forms the basis for the analytical design of blade arrangements in axial flow machines, for which the volume or mass flow rate, the pressure build-up at a given working fluid density as well as the assumed rotational speed are the input data for the calculation.

Available online by August 31st, 2023 https://doi.org/10.15866/irease.v16i4.24065

Figure 1 shows the geometry of the rectilinear blade cascade for the case of flow through a rotating component, together with the velocity triangle system.

Many algorithms for the calculation of axial blade arrangements in fluid-flow machines based on the rectilinear blade cascade can be found in the literature.

The main literature references include [1]-[3]. The rectilinear blade cascade model has undoubted advantages in terms of the simplicity of describing the kinematics of the flow through the stages of an axial flow machine and the feasibility of estimating the characteristics of the designed flow machine. However, it also has disadvantages, such as the infeasibility of estimation of flow losses or the infeasibility of considering aerodynamic phenomena that are negative from the point of view of blade cascade performance, such as the detachment of the boundary layer of flow.

This model also does not take into account the interference between the rotor stage and the stator stage.

Regardless of its disadvantages, the rectilinear blade cascade model is an effective tool for preliminary design engineering of axial flow machines. However, in order to use this model, the knowledge of the aerodynamic characteristics of the airfoils in a blade cascade arrangement is required. In [4], it has been shown that when designing axial flow machines, the values for the lift and drag force coefficients for isolated airfoils cannot be used in the case of airfoils in a blade cascade arrangement. This has been also discussed in [5] for the NACA 0012 airfoil in a blade cascade and isolated arrangement. The values of these coefficients for isolated airfoils and airfoils in a blade cascade arrangement are quite different. The waveforms (trends) of the characteristics of $C_l=f(\alpha)$ and $C_d=f(\alpha)$ very between the two airfoil configurations. Values for the aerodynamic coefficients for the airfoil arrangement in the blade cascade can be obtained from the relevant reference data sheets. However, to these authors' knowledge, the only publicly available data sheet that contains aerodynamic characteristics of airfoils in a rectilinear blade cascade arrangement is the NACA REPORT 1368 [6]. However, this catalogue is limited only to the NACA 65 series airfoils and a few configurations of flow intake angles and blade cascade solidities. With a dedicated wind tunnel, it is possible to determine experimentally the aerodynamic characteristics of the airflows in a blade cascade arrangement.

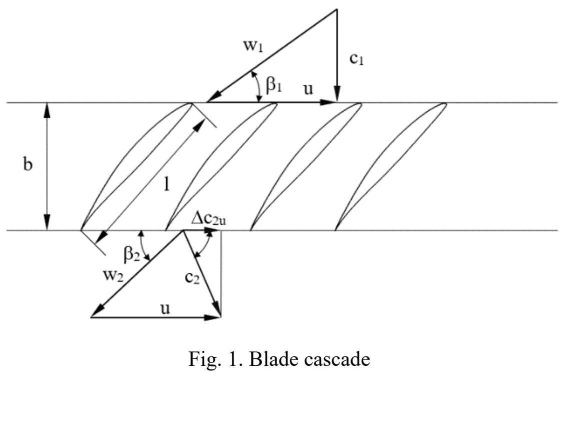

Fig. 1. Blade cascade

The results of this type of test are shown in [7] and [8]. However, this entails a sophisticated testing infrastructure. At the same time, due to the cost, length of measurements, multiplicity of configurations and number of airfoil families, the experimental determination of airflow characteristics in a blade cascade arrangement is not a universal method. Hence, an excellent solution to the problem of determining blade airfoil characteristics in a blade cascade arrangement is to carry out numerical flow simulations. However, there are also specific problems in this aspect. It is difficult to find data on the aerodynamic coefficients of airfoils in blade cascade arrangements resulting from flow simulations in scientific papers. K. M. Pandey et al. [9] have presented an approach for developing models for performing numerical simulations of flow through a rectilinear blade cascade, building numerical meshes, as well as carrying out calculations by using periodic boundary conditions.

However, the analysis of the results performed only focuses on the pressure, the velocity, and the temperature fields. Obviously, this is important in terms of assessing the correct positioning of the airfoils in the blade cascade for the preset operating parameters. As the desktop review shows, with regard to the distribution of velocity fields determined experimentally and numerically, there is good agreement between the results obtained [10], even for supersonic flows [11], [12]. However, such an analysis has been required to optimise the blade cascade arrangement of airfoils for increased performance. As shown in the literature on the subject, the numerical solution of flow through a blade cascade can be successfully carried out by using the finite element method [13] as well as the panel method [14]. The issue of analysing velocity and pressure fields in a flow through a blade cascade is also shown in [15] and [16].

These sources indicate that numerical simulations are useful in the process of optimising airfoils for use in blade cascades. The analyses carried out have showed the positive effect of geometry optimisation on flow loss reduction. [17] discusses an approach for resolving potential flow through a blade cascade. The mathematical formulas presented in this paper, based on the Kutta condition and the Riemann-Hilbert approach, allow the velocity distribution around the airfoils in the cascade arrangement to be determined, along with the circulation distribution, which can be combined with the lift coefficient. The approach presented in [17] is quite promising for the rapid analytical determination of aerodynamic characteristics of airfoils in a blade cascade, as also mentioned in [18]. In addition, the analytical formulation of the blade cascade flow problem also allows transient issues to be solved [19]. The approach shown in [19] concerns the determination of flow fields without going into the aerodynamic coefficients of airfoils. Numerical analysis of flow through a blade cascade finds application in the broader analysis of flow loss. These losses primarily affect the values of the total pressure rise generated and the efficiency of the flow machine. An approach to analyse such issues is included in [20]. [21] shows a comparison of numerical results with experimental results of flow through a set of blade cascades.

The results show that the author has obtained a satisfactory agreement of the pressure loss distribution without going into the forces arising on the blades, enabling the determination of the coefficients C_l and C_d.

Mathematical formulas for the forces applied to the airfoils in a blade cascade arrangement are also provided in [22], where they have been extensively analysed. By analysing the reference literature, many papers can be found concerning the analysis of flow through a rectilinear blade cascade – with analytical [23], numerical or experimental [24] methods. Unfortunately, the results have reported mainly concern pressure and velocity distributions. It is difficult to find the results of numerical analyses of the aerodynamic coefficients C_l and C_d and their comparison to experimental results. This also applies to the flow-turning angle of the working medium flowing through the blade cascade. Knowing the value of the flow-turning angle is extremely useful in the design process of axial flow machines. This paper presents a method for the numerical determination of the aerodynamic coefficients C_l and C_d of airfoils in a blade cascade arrangement and the flow turning angle of air through a rectilinear blade cascade. The focus has been on a blade cascade based on the NACA 65-010 airfoil with a cascade solidity σ=1 and flow intake angle β_1=30°. First, the computational domain model has been discussed, including its dimensions, which has been then discretized by using the so-called numerical hybrid meshes. 17 numerical meshes have been built, differing in the total number of elements as well as the number of layers of rectangular elements in the area of the boundary layer. A series of numerical flow simulations have been performed for the prepared meshes in order to determine the influence of the mesh on the simulation results. Then, a numerical analysis of the impact of the turbulence model on the results of numerical simulations was presented. For this purpose, the characteristics $C_l=f(\alpha)$, $C_d=f(\alpha)$ and $\theta=f(\alpha)$ of the NACA 65-010 blade cascade with previously mentioned parameters have been numerically reproduced. The numerical results have been compared to experimental data from the report [6]. Flow simulations have been carried out in Ansys Fluent software by solving RANS equations, a method used successfully to solve various aerodynamics problems in [25] to [29].

II. Computational Domain and Boundary Condition

In this paper, an attempt has been made to replicate the experimental study of a rectilinear blade cascade [6] using numerical analyses. To this end, a suitable blade cascade model has been built to carry out numerical flow simulations. This model is shown in Figure 2. As already mentioned, the NACA 65-010 airfoil has been studied in the blade cascade arrangement.

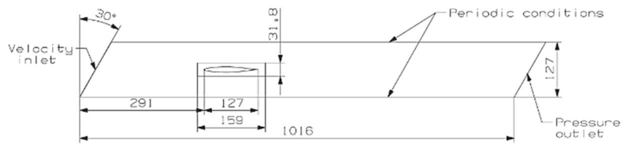

Fig. 2. Computational domain and boundary conditions

The computational domain is a parallelogram with 1016 mm × 127 mm sides. The acute angle of this parallelogram has been 60°, which is because the model has been prepared for a flow intake angle value of 30°.

Inside the parallelogram, an additional rectangle of 159 mm × 31.8 mm has been built to make the numerical mesh more dense in close proximity to the 127 mm chord airfoil. The chord value has corresponded to the chord of the airfoil used in the experimental study [6]. To the left edge of the parallelogram (Fig. 2), a velocity-inlet type condition has been assigned, while a pressure-outlet type condition has been assigned to the right edge. Periodic boundary conditions have been assigned to the lower and upper edges of the parallelogram representing the computational domain, thus obtaining an infinite flat blade cascade of airfoils. The model prepared has been processed by digitising. It has been decided to build a hybrid mesh. In the boundary layer area, layers of rectangular elements have been modelled, and the remaining area has been digitised by using triangular elements. In total, a set of 17 numerical meshes have been built, by varying in the number of elements, the number of layers of prismatic elements and the height of the prismatic elements, resulting from the chosen value of the parameter ‘wall y+’. This approach has been used to analyse the influence of the mesh parameters on the results of the numerical simulations. These are discussed later in this article. Figure 3 shows an example of a numerical mesh. For the suitably prepared numerical meshes, a series of simulations have been run with the following flow conditions:

- Velocity-inlet: 28.956 m/s;
- Pressure-outlet: 0 Pa;
- Reference temperature: 300 K;
- reference pressure: 101325 Pa.

III. Influence of the Meshes on Numerical Results

In order to analyse the influence of the mesh parameters on the results of the numerical simulations, the case of flow around a blade cascade composed of NACA 65-010 airfoils has been selected for the flow intake angle β_1=30° and the angle of attack α=3°. It has been assumed that numerical simulations would be performed for the Spalart-Allmaras (S-A) turbulence model. Figure 4 shows a comparison of the lift coefficients obtained by using the numerical method, with a variable number of mesh elements, and the experimental method. The meshes contained no prismatic elements in the boundary layer region, and the determined wall y+ value varied between 1.77÷26.65, with smaller wall y+ values obtained for meshes with more elements. Based on the results obtained for the lift coefficient, a negligible effect of the number of mesh elements on the values C_l could be recognised. The numerically obtained values of the lift coefficient for meshes differing in the number of elements are in the range 0.073 – 0.076. Values of the C_l coefficient closer to the experimental value have been obtained for meshes with a larger number of elements. At the same time, it is important to state that satisfactory agreement has been obtained between the numerical and experimental results.

For meshes characterised by a preset number of elements (Fig. 4) and the previously mentioned wall y+ values, the drag force coefficient has been also read, as shown in Figure 5. In this case, it is clearer that for meshes characterised by a smaller number of elements (< 100,000), the values C_d significantly exceed the value of the drag force coefficient obtained in the experiment.

For meshes below the number of elements 100,000, the obtained values of the C_d coefficient are close to the value of 0.015 with the value of 0.0126 obtained in the experiment.

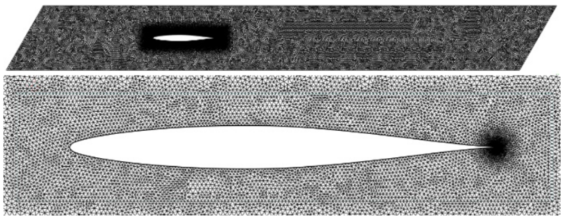

Fig. 3. A numerical mesh example

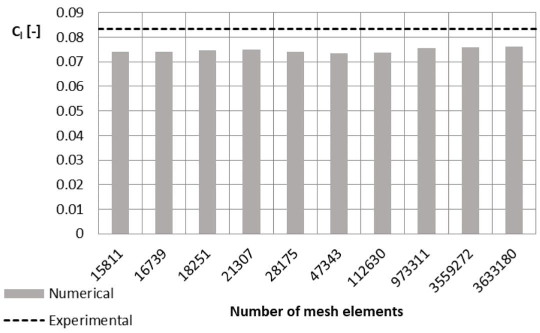

Fig. 4. Lift coefficient values as a function of the number of mesh elements

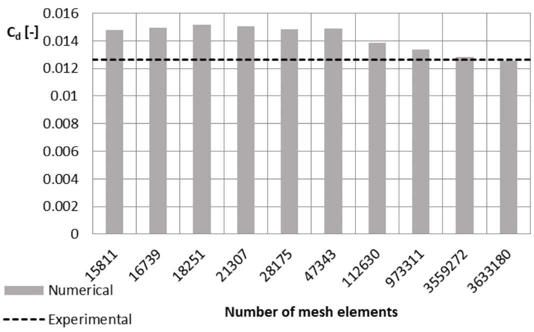

Fig. 5. Drag coefficient values as a function of the number of mesh elements

For meshes with more than 100,000 elements, the values C_d have been close to the value obtained experimentally in the wind tunnel [6], while it should be recognised that for the 2D case and the developed model, meshes with more than 1,000,000 elements have been already very large in the context of the numerical tests performed. Figure 6 shows the results of the flow-turning angle of air through the blade cascade model. In [6], the flow turning angle has been measured with a claw-type yaw head at a distance of 1 to 3 chord values of the airfoil behind the airfoil. For the numerical simulations, it has been decided to read the value of the flow turning angle at a distance of two chords from the trailing edge of the airfoil. In the case of the airflow turning angle, it is clear that the numerically obtained values have deviated significantly from the experimentally obtained value. It should also be noted that the number of mesh elements have not significantly affected the readings of the flow turning angle.

The values of the flow turning angle, determined numerically, range from approximately 1.77° to 1.88°.

According to [6], for the angle of attack $\alpha = 3^o$, the airflow turning angle has been 2.37°. The cases analysed above should be considered as absolute differences between the experimental and numerical results. In order to identify properly differences in the results, relative differences should be considered. Table I lists the values of the relative differences of the C_l, C_d and θ results obtained numerically compared to the experimental results [6].

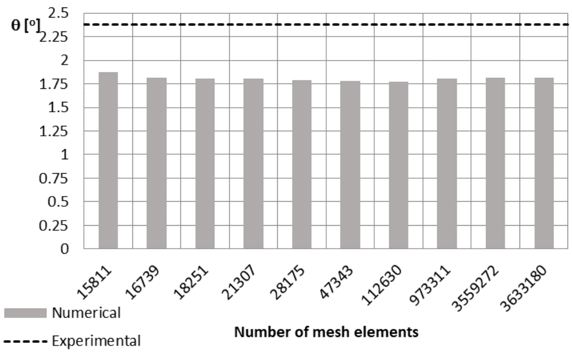

Fig. 6. Turning angle values as a function of the number of mesh elements

TABLE I
RELATIVE DIFFERENCES IN NUMERICAL VALUES OF C_l, C_d AND θ FROM EXPERIMENTAL RESULTS FOR DIFFERENT NUMBERS OF MESH ELEMENTS

Number of mesh elements [-]	δC_l [%]	δC_d [%]	$\delta\theta$ [%]
15811	11.24	17.12	20.94
16739	11.21	18.37	23.74
18251	10.41	20.14	23.95
21307	10.22	19.51	24.03
28175	11.02	17.58	24.67
47343	11.77	18.19	24.91
112630	11.71	9.65	25.32
973311	9.35	5.98	23.89
3559272	8.81	1.78	23.78
3633180	8.58	0.30	23.77

From the differences in numerical versus experimental results in Table I, it can be seen that as the number of mesh elements increases, the relative differences in the values obtained C_l, C_d decrease. The smallest discrepancies have been obtained for a mesh with 3,633,180 elements. For the numerical mesh constructed in this way, the difference in the lift force coefficient results obtained numerically and experimentally has amounted to 8.58%. In the case of the drag coefficient, the difference in results is only 0.3%. This has not applied to the value of the airflow turn angle, where, irrespective of the number of mesh elements, the differences between the experimental and numerical results have ranged from 21% to 24%. In the next step, it has been decided to analyse the impact of the number of layers of rectangular elements in the boundary layer area on the results of the numerical simulations. For this type of analysis, a mesh of 112,630 elements has been built, for which 1 to 7 layers of rectangular elements have been modelled. A mesh of this number of elements has been chosen because of its reasonable size in the context of 2D simulation for the chosen size of the computational domain (Fig. 2). Obviously, this approach has increased the number of mesh elements slightly, with the resulting wall y+ value being less than 1 for all cases considered.

A set of charts comparing the numerical and experimental results of lift coefficient, drag coefficient and flow turning angle as a function of the number of layers of rectangular elements is shown in Figs. 7-9.

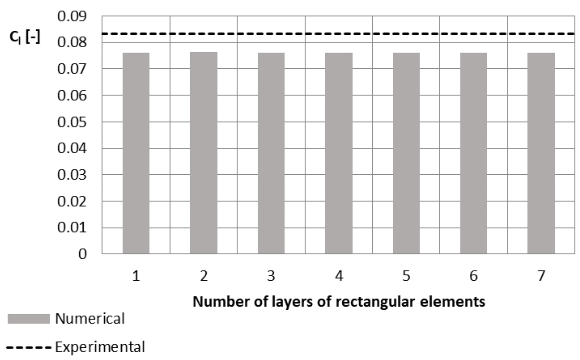

Fig. 7. Lift coefficient values as a function of the number of layers of rectangular elements

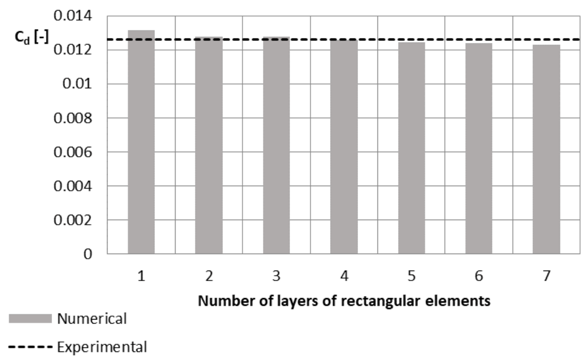

Fig. 8. Drag coefficient values as a function of the number of layers of rectangular elements

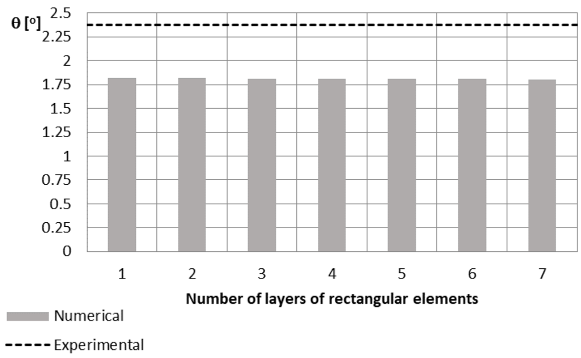

Fig. 9. Turning angle values as a function of the number of layers of rectangular elements

From the results shown in Figs. 7-9, it can be deduced that as the number of layers of rectangular elements in the boundary layer area has increased, the difference in the C_d values obtained numerically and experimentally has decreased.

It appears that for the rectilinear blade cascade case in question, the optimum solution would be to create 4 layers of prismatic elements. With these mesh settings, the numerical and experimental results have been almost identical. In the case of the results of the lift force coefficient and the flow turning angle, it can be recognised that there has been no influence of the number of layers of rectangular elements in the boundary layer area on the values of the results obtained. It can be stated that regardless of the number of rectangular elements in the area of the boundary layer, the values of the lift coefficient and the turning angle, obtained numerically, remain unchanged. The above statements are confirmed by the absolute difference values of the experimental and numerical results in Table II. Similar flow simulations have been performed for meshes characterised by the desired wall y+ 30 and 60.

However, the obtained values of C_l, C_d and θ have been far from the experimental values. Moreover, an additional adaptation in the boundary layer area, consisting of an additional subdivision of the rectangular elements in this area, has not improved the numerical results. Hence, they are not presented in this article. For all the calculation cases analysed above, the differences between the experimental and the numerical results are primarily to be found in the experimental method discussed in [6].

TABLE II
RELATIVE DIFFERENCES IN NUMERICAL VALUES OF C_l, C_d AND θ FROM EXPERIMENTAL RESULTS FOR DIFFERENT NUMBERS OF MESH ELEMENTS

Number of prismatic layers [-]	δC_l [%]	δC_d [%]	δθ [%]
1	8.70	4.27	23.52
2	8.49	1.30	23.57
3	8.63	1.13	23.70
4	8.52	0.20	23.74
5	9.03	0.61	24.05
6	8.62	1.85	23.95
7	8.57	2.58	23.98

Among other things, this method does not take into account the degeneration of C_l values due to the viscosity of the fluid. The numerical method has taken into account the influence of viscosity on the values of the aforementioned aerodynamic coefficients according to the chosen turbulence model.

IV. Influence of the Turbulence Models on Numerical Results

After positively verifying the influence of the mesh parameters on the results of numerical simulations of flow through a rectilinear blade cascade, it has been decided to determine the characteristics of the NACA 65-010 airfoil cascade for flow intake angle β_1=30° and solidity σ=1, by using a mesh with a total of 112,630 elements and 4 layers of rectangular elements in the boundary layer area. Flow simulations have been carried out in the range of attack angles α=-4° to 14° for the following settings:

- Solver options: double – precision;
- Solver type: Pressure – Based;
- Time: steady;
- Turbulence models: Spalart – Allmaras (S-A) and SST k-ω.

The experimental values have been read for the angles of attack given in [6]. Two turbulence models have been chosen to test the impact of their application on the results of the numerical simulations. Both the S-A model and the SST k-ω model are widely applied in the numerical solution of aerodynamics of flying objects and turbine machines [30], [31]. Figure 10 shows a comparison of the characteristics $C_l=f(\alpha)$ obtained experimentally and numerically. From the characteristics shown, there is a satisfactory agreement between the numerical and the experimental results, especially in the low angle of attack range. In the range of attack angles α=0° to 6°, the numerical results almost coincide with the experimental results. It should be noted here that airfoils in the rectilinear blade cascade operate at relatively low attack angles, which prevents the phenomenon of detachment of the boundary layer. The occurrence of detachment of the boundary layer during the operation of an axial flow machine may lead to unstable operation. As the angle of attack has increased, the numerical results have begun to diverge more significantly from the numerical results. Differences can also be seen in the slopes of the $C_l=f(\alpha)$: the experimental curve had a smaller slope angle than those obtained numerically. This could be because the curvature of the airfoils used in the experiment has differed from those mapped in CAD for the numerical simulations.

Differences in experimental and numerical results for $C_l=f(\alpha)$ could also stem from the accuracy of the reading of the experimental values. There has been no significant effect of the adopted turbulence models on the results of the numerical simulations. The values of $C_l=f(\alpha)$ for both turbulence models have coincided. In addition, Table III shows the numerical values of the lift coefficient.

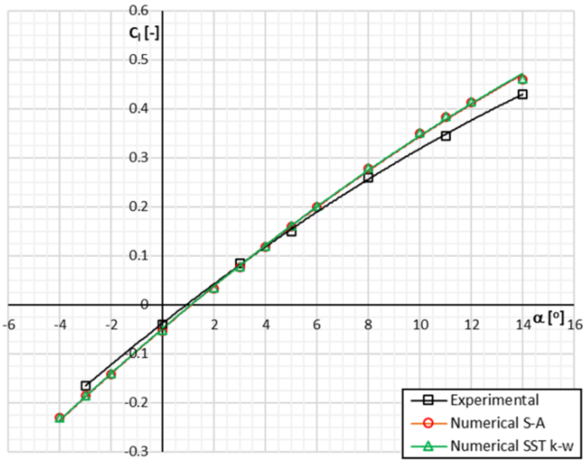

Fig. 10. Characteristics of $C_l=f(\alpha)$ NACA 65-010 airfoil cascade for β_1=30° and σ=1

TABLE III
NUMERICAL VALUES OF THE LIFT COEFFICIENT OBTAINED IN THE EXPERIMENT AND NUMERICALLY

α [°]	C_{l_exp} [-]	$C_{l\,S\text{-}A}$ [-]	$C_{l\_SST\,k\text{-}\omega}$ [-]
-3	-0.165	-0.1860	-0.1854
0	-0.04	-0.0531	-0.0525
3	0.085	0.0762	0.0770
5	0.15	0.1595	0.1605
8	0.26	0.2777	0.2786
11	0.345	0.3829	0.3836
14	0.43	0.4585	0.4609

Figure 11 shows a comparison of the characteristics of $C_d=f(\alpha)$ obtained experimentally and numerically. In this case, almost identical results to those reported in [6] have been obtained by using the S-A turbulence model. This applies to the entire range of angles of attack considered.

In the case of the SST k-ω turbulence model the drag force coefficient values obtained have been underestimated compared to the experimental and numerical data from the S-A model. It can be concluded that the characteristic $C_d=f(\alpha)$ obtained using the SST k-ω turbulence model is shifted in parallel towards lower C_d values compared to the characteristics determined for the S-A turbulence model. Therefore, it can be stated that when determining the characteristics of the drag coefficient of airfoils in the rectilinear blade cascade, calculations can be made by using the S-A turbulence model, provided that relatively small angles of attack and no detachments of the boundary layer are involved. Table IV shows the numerical values of the drag force coefficient obtained from the numerical tests and read from [6]. The last set of characteristics determined has been those of flow turning angle as a function of the angle of attack. This angle has been read at a distance of two chords from the trailing edge of the airfoil. The characteristics are shown in Figure 12. In this case, too, satisfactory agreement between the experimental and the numerical results has been obtained, with the numerically obtained characteristics of $\theta=f(\alpha)$ being virtually parallel-shifted, relative to the experimental characteristics, towards lower values of the flow turning angle.

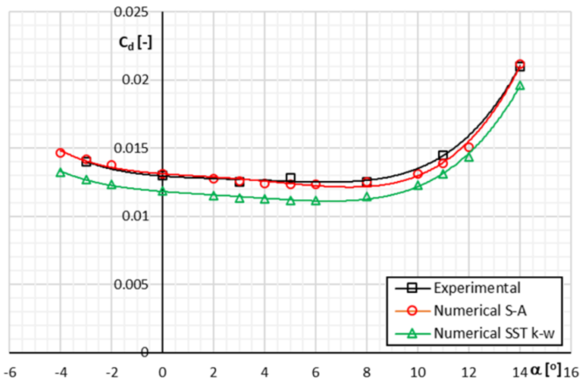

Fig. 11. Characteristics of $C_d=f(\alpha)$ NACA 65-010 airfoil cascade for $\beta_1=30°$ and $\sigma=1$

TABLE IV
NUMERICAL VALUES OF THE DRAG COEFFICIENT OBTAINED IN THE EXPERIMENT AND NUMERICALLY

α [°]	C_{d_exp} [-]	$C_{d\_S\text{-}A}$ [-]	$C_{d\_SST\ k\text{-}\omega}$ [-]
-3	0.014	0.01417	0.01268
0	0.013	0.01313	0.01185
3	0.0125	0.01259	0.01132
5	0.0128	0.01236	0.01116
8	0.0125	0.01251	0.01145
11	0.0145	0.01388	0.01312
14	0.021	0.02116	0.01963

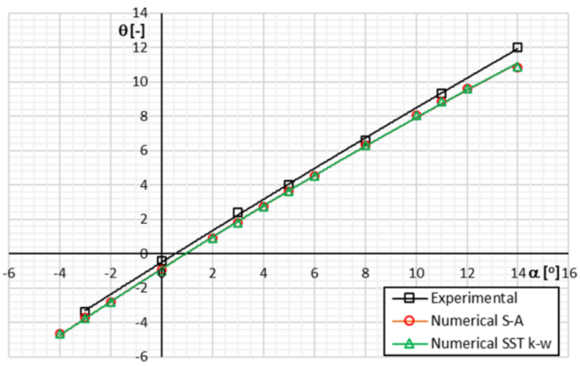

Fig. 12. Characteristics of $\theta=f(\alpha)$ NACA 65-010 airfoil cascade for $\beta_1=30°$ and $\sigma=1$

TABLE V
NUMERICAL VALUES OF THE DRAG COEFFICIENT OBTAINED IN THE EXPERIMENT AND NUMERICALLY

α [°]	$\theta_{\_exp}$ [°]	$\theta_{\_S\text{-}A}$ [°]	$\theta_{\_SST\ k\text{-}\omega}$ [°]
-3	-3.4	-3.73	-3.74
0	-0.4	-0.95	-0.96
3	2.4	1.81	1.81
5	4	3.64	3.64
8	6.6	6.32	6.32
11	9.3	8.84	8.84
14	12	10.84	10.87

From the presented characteristics $\theta=f(\alpha)$, this shift is characterized by a constant value over the entire range of analyzed angles of attack. There has been no significant effect of the adopted turbulence model on the flow turning angle values obtained. The characteristics of $\theta=f(\alpha)$ have coincided between the S-A and SST k-ω turbulence models. Table V shows the numerical values of the flow turning angle obtained in the numerical tests and read from [6].

V. Conclusion

This paper presents a methodology for the numerical determination of aerodynamic characteristics of airfoils in a rectilinear blade cascade arrangement. The methodology is demonstrated by using the example of a blade cascade formed by NACA 65-010 airflows for flow intake angle $\beta_1=30°$ and blade cascade solidity $\sigma=1$. An approach to building a blade cascade model and an analysis of the effect of hybrid numerical mesh parameters and turbulence models on flow simulation results is shown. The obtained values for lift coefficient, drag coefficient and flow turning angle have been compared to the experimental data provided in [6].

In terms of the analysis of the influence of mesh parameters on the results of numerical simulations, a significant effect of the number of mesh elements on the drag force coefficient values obtained has been demonstrated. It can be concluded that, for properly modelled meshes, the number of elements does not affect the numerically determined values of lift coefficient and flow turning angle. The modelling of rectangular elements in the boundary layer area has a greater impact.

However, it should be noted, as demonstrated in the body of this paper, that an increase in the number of layers of rectangular elements does not necessarily translate into an increase in the accuracy of the results obtained. In terms of the influence of turbulence models, it has been shown that for the airfoil characteristics in the blade cascade, $C_l=f(\alpha)$ and $\theta=f(\alpha)$, regardless of the S-A or SST k-ω model used, the results have not been significantly different. In the case of the $C_d=f(\alpha)$ characteristics, it has been shown that the S-A turbulence model has allowed numerical results almost identical to experimental results. The use of the SST model k-ω has resulted in a significant underestimation of the drag force coefficient obtained compared to experimental data.

In summary, it is possible to determine numerically the aerodynamic characteristics of blade cascade airfoils for the design process of axial flow machines. Geometrically simple blade cascade models and meshes characterised by a relatively small number of elements can be used successfully for this purpose. The study shows that it would be sufficient to create 3 to 5 layers of such elements. At the same time, the Spalart-Allmaras single-equation turbulence model can be used for the numerical determination of airfoil characteristics in a rectilinear blade cascade arrangement consistent with experimental data.

References

[1] E. Tuliszka, *Compressors, blowers and fans* – in polish, WNT, Warsaw, 1976.

[2] A. Witkowski, *Rotary compressors. Theory, construction, operation* – in polish (Publishing House of the Silesian University of Technology, Gliwice, 2013).

[3] T. Wright, *Fluid machinery: performance, analysis and design*, CRC Press, Florida, 1999.

[4] M. Majcher, *Numerical analysis of three-dimensional flows in key components of axial fans* – in polish, Ph.D. dissertation, Military University of Technology, Warsaw, 2021.

[5] N. Ahmed, B.S. Yilbas, M. O. Budair, Computational study into the flow field developed around a cascade of NACA 0012 airfoils, *Computer Methods in Applied Mechanics and Engineering,* 167, (1998), 17-32.
doi: 10.1016/S0045-7825(98)00104-2

[6] J. C. Emery, L. J. Herrig, J. R. Erwin, A. R. Felix, *REPORT 1368 Systematic Two-dimensional Cascade Tests of NACA 65-series Compressor Blades at Low Speeds*, (Work of the US Gov., Langley Aeronautical Laboratory Langley Field, Va., 1951).

[7] S. Panchal, V. Mayavanshi, Experimental study of flow through compressor Cascade, *Case Studies in Thermal Engineering,* Volume 10 (2017), 234-243.
doi: https://doi.org/10.1016/j.csite.2017.05.002

[8] N. G. Rodionov, Experimental Studies of Airfoil Cascades with High Velocity Vector Circulation around the Airfoil, *Thermal Engineering,* volume 69, (2022), 42-50.
doi: 10.1134/S0040601521110033

[9] K.M. Pandey, S.Chakraborty, K.Deb, CFD Analysis of Flow through Compressor Cascade, *International Journal of Soft Computing and Engineering (IJSCE),* Volume-2, Issue-1 (2012), 362-371.

[10] D. Engelmann, M. Sinkwitz, F. di Mare, B. Koppe, R. Mailach, J. Ventosa-Molina, J. Fröhlich, T. Schubert, R. Niehuis, Near-Wall Flow in Turbomachinery Cascades - Results of a German Collaborative Project, *International Journal of Turbomachinery, Propulsion and Power,* 6, 9, (2021).
doi: https://doi.org/10.3390/ijtpp6020009

[11] P. Louda, P. Straka, J. Příhoda, Simulation of transonic flows through a turbine blade cascade with various prescription of outlet boundary conditions, *EPJ Web of Conferences*, 180, 02056 (2018).
doi: 10.1051/epjconf/201818002056

[12] P. Louda, J. Příhoda, P. Šafařík, investigation of compressible flow througha turbine blade cascade for various transonic flow regimes, *Acta Polytechnica,* 61, (2021), 110–116,
doi: https://doi.org/10.14311/AP.2021.61.0110

[13] M. Mesbah, V. G. Gribin, K. Souri, Evaluation of different turbulence models in simulating the subsonic flow through a turbine blade cascade, *IOP Conf. Series: Materials Science and Engineering* 1092, (2021).
doi: 10.1088/1757-899X/1092/1/012064

[14] Z.Lei and G.Liang, Solution of Turbine Blade Cascade Flow Using an Improved Panel Method, *International Journal of Aerospace Engineering,* Article ID 312430, (2015), 6 pages.
doi: http://dx.doi.org/10.1155/2015/312430

[15] F. Meng, C. Gong, K. Li, J. Xiong, J. Li, P Guo, Aerodynamic Optimization and Mechanism Investigation on Performance Improvements in a Transonic Compressor Cascade, *Machines,* 11(2):244, (2023)
doi: https://doi.org/10.3390/machines11020244

[16] S. Huanga, X. Lua, G. Hana, S. Zhao, C. Zhou, C. Yang, Research on aerodynamic optimization design method and flow mechanism of ahigh-subsonic compressor cascade, *Engineering Applications of Computational Fluid Mechanics,* Volume 16, (2022).
doi: https://doi.org/10.1080/19942060.2021.2020170

[17] P. J. Baddoo, L. J. Ayton L. J. Potential flow through a cascade of aerofoils: direct and inverse problems, *Proceedings of the Royal Society A: Mathematical, Physical and Engineering Sciences* 474, (2018).
doi: 10.1098/rspa.2018.0065

[18] Nutanapati S., Perspective on Potential Flow through a Cascade of Aerofoils, *Journal of Aeronautics & Aerospace Engineering* Vol. 10 Iss. 8 No: 265, (2021).

[19] E. G. Ladopoulos, Non-linear singular integral equations analysis for unsteady cascade aeroelasticity applied in turbomachines, *Archives ff Mechanics,* 65, 1, (2013), 45–53.

[20] T. Zheng, X. Qiang, J. Teng, J.Feng, Numerical Loss Analysis In A Compressor Cascade With Leading Edge Tubercles, *Journal Of Theoretical And Applied Mechanics,* 56, 4, (2018), 1083-1095.
doi: 10.15632/jtam-pl.56.4.1083

[21] G. L. Martins, Axial Turbine Cascade Correlation, *Applied Sciences,* 6, 420, (2016).
doi: 10.3390/app6120420

[22] Narges Golmirzaee, David Wood; Investigating horizontal-axis wind turbine aerodynamics using cascade flows. *J. Renewable Sustainable Energy* 1 July 2023; 15 (4): 043302.
doi: https://doi.org/10.1063/5.0147946

[23] A. R. Kriebel, *Stall Propagation in a cascade of airfoils, Gas Turbine Laboratory*, (Massachusetts Institute Of Technology, Cambridge, Report No. 36, 1956).

[24] K. Funazaki, K. Yamada, T. Ono, K. Segawa, H. Hamazaki, A. Takahashi, H. Tanimitsu, Experimental and Numerical Investigations of Wake Passing Effects upon Aerodynamic Performance of a LP Turbine Linear Cascade With Variable Solidity, *Asme Turbo Expo,* (2006).

[25] Czyż, Z., Karpiński, P., Skiba, K., Wendeker, M., Measurements of Aerodynamic Performance of the Fuselage of a Hybrid Multi-Rotor Aircraft with Autorotation Capability, (2022) *International Review of Aerospace Engineering (IREASE)*, 15 (1), pp. 12-23.
doi: https://doi.org/10.15866/irease.v15i1.21319

[26] Almawla, A., Lateef, A., Kamel, A., Water Flow Simulation with Computational Fluid Dynamics (CFD): a Review Study, (2022) *International Review of Civil Engineering (IRECE)*, 13 (1), pp. 40-52.
doi:https://doi.org/10.15866/irece.v13i1.20958

[27] Rojas, J., Valencia Ochoa, G., Duarte Forero, J., CFD Analysis of Swirl Effect in a Diesel Engine Using OpenFOAM, (2020) *International Review on Modelling and Simulations (IREMOS)*, 13 (1), pp. 8-15.
doi:https://doi.org/10.15866/iremos.v13i1.18372

[28] Suranto Putro, S., Sutardi, S., Widodo, W., Pambudiyatno, N., Sonhaji, I., Effect of Leading-Edge Gap Size on Multiple-Element Wing NACA 43018, (2022) *International Review of Aerospace Engineering (IREASE)*, 15 (6), pp. 321-331.
doi: https://doi.org/10.15866/irease.v15i6.22664

[29] Kozakiewicz, A., Frant, M., Majcher, M., Impact of the Intake Vortex on the Stability of the Turbine Jet Engine Intake System, (2021) *International Review of Aerospace Engineering (IREASE)*, 14 (4), pp. 173-180.
doi: https://doi.org/10.15866/irease.v14i4.20223

[30] Mahboub, A., Bouzit, M., Ghenaim, A., Effect of Curvilinear and Inverted Aircraft Spoiler Deflection Angle on Aerodynamic Wing Performances, (2022) *International Review of Aerospace Engineering (IREASE)*, 15 (3), pp. 151-161.
doi: https://doi.org/10.15866/irease.v15i3.21317

[31] Shivaramaiah, S., Varpe, M., Aerodynamic Performance and Stability of a Transonic Axial Compressor Stage with an Airfoil Vortex Generator, (2023) *International Review of Aerospace Engineering (IREASE)*, 16 (3), pp. 110-122.
doi: https://doi.org/10.15866/irease.v16i3.23569

Authors' information

Military University of Technology (MUT), Faculty of the Mechatronics, Armament and Aerospace.

Maciej Majcher (Corresponding Author), PhD Eng, was born in 1989 in Warsaw, Poland. In 2008, he obtained an engineer title at the Military University of Technology. In 2013 he obtained a master's degree in airplanes and helicopters specialty. Since 2014, he has been working at the Military University of Technology, at the Faculty of Mechatronics, Armament and Aviation, at the Institute of Aviation Technology, at the Department of Aerodynamics and Thermodynamics. In 2021, he obtained a PhD in technical in the field of three-dimensional flow analysis in key elements of axial fans. He is a co-author of 21 articles, one monograph and an expert opinion. He deals with issues related to flows through axial flow machines, design of flow machines with a

focus on fans, computer-aided design and manufacturing processes and generally understood computer fluid mechanics.
E-mail: maciej.majcher@wat.edu.pl

Michał Frant, PhD Eng, was born in Łask, 1975, holds a master's degree (Military University of Technology, Warsaw, Poland, 1999) in mechanical engineering, airplanes and helicopters specialty, doctorate in Mechanics (Military University of Technology, Warsaw, Poland, 2010) fluid mechanics specialty. His research interests include CFD analysis, experimental and numerical aerodynamics, axial fan performance. He has 25 years of experience in these areas. Dr Frant is an assistant professor at Faculty of the Mechatronics, Armament and Aerospace, Military University of Technology. He is a co-author of 34 articles and one monograph.

Rafał Kieszek, PhD Eng., was born in 1992 in Wolomin, Poland. In 2015, he obtained an engineer title and in 2016 a master's degree in airplanes and helicopters specialty (Military University of Technology). Since 2016, he has been working at the Military University of Technology at the Faculty of Mechatronics, Armament and Aviation, at the Institute of Aviation Technology. In 2023, he obtained a PhD in technical in the field of flow machines. He is a co-author of 6 articles. He deals with issues related to the design and optimization of flow machines using genetic algorithms and machine learning.