This paper is available online at www.praiseworthyprize.org
International Review of Aerospace Engineering (I.RE.AS.E), Vol. 14, N. 4
ISSN 1973-7459 August 2021

Impact of the Intake Vortex on the Stability of the Turbine Jet Engine Intake System

Adam Kozakiewicz, Michał Frant, Maciej Majcher

Abstract – *The article presents a numerical analysis of the intake system of a turbine jet engine in terms of parameter stability along its duct, following the occurrence of an intake vortex. This type of intake system is characterized by high susceptibility to intake vortex. In extreme cases, this type of phenomenon leads to the engine surge and even to the operation disruption (engine stalling). The article presents a developed model of the front part of the aircraft with an intake duct. The discretization process involved in the issue under consideration has been described. The airflow parameters corresponding to the conditions in such cases have been adopted and numerical calculations have been performed. The result is an intake vortex. Subsequently, significant cross sections in the intake system have been separated, on which the impact pressure distributions have been determined. The main part of the article is devoted to the analysis of pressure distributions. They have been subjected to quantitative analysis using the proposed pressure coefficient. The coefficient has provided quantitative information about the difference in pressure distributions for selected sections. The results obtained have provided information about mounting airflow instability in the flow duct caused by the intake vortex.*

***Keywords**: Intake Vortex, Numerical Fluid Dynamics, Turbine Jet Engine, Jet Engine Inlet, Pressure Distribution in the Engine Inlet*

Nomenclature

A_1	Final cross-section of the engine intake duct
A_H	Free cross-airflow section
A_i	Dimensionless area in the ith area
c_v	Specific heat at constant volume
$C_{x,wl}$	Intake drag coefficient
$DC(\Theta)$	Distortion coefficient
k	Isentrope exponent
ΔK_{wl}	Surge margin
Ma	Mach number
n	Number of separate pressure areas
$\bar{n}_s$	Deviation of engine rotor speed changes
$\bar{N}_s$	Deviation of the turbine power changes
$p*$	Total pressure
p_i	Average total pressure in the ith area
$\dot{q}_n$	Surface density of the heat flux (e.g. Fourier law for heat conduction $\dot{q}_n = \lambda \vec{n} gradT$)
$\dot{q}_m$	Heat flux density related to the unit of mass of the fluid
$q(\lambda)$	The relative airflow density
R	Gas constant
S	Area
t	Time
T	Temperature
$T*$	Total temperature
$\bar{T}_4$	Deviation of turbine outlet temperature changes
V	Volume
λ	Thermal conductivity
v	Velocity
π^*_{wl}	The intake pressure ratio
ρ	Density
σ^*_{wl}	Pressure loss coefficient
φ_{wl}	Flow rate coefficient

I. Introduction

The basic task of a turbine jet engine is to ensure adequate thrust while meeting additional criteria related to the entire engine as well as its components (inlet - compressor - combustion chamber - turbine - exhaust system). For this reason, a number of works are carried out to optimize individual components, as well as to determine the optimal conditions for their cooperation, including the inlet-compressor cooperation.

The intake system of a jet engine is often a critical component of an engine's performance and operational reliability. The correct operation of the intake system also affects the performance of the entire aircraft, e.g. its maneuverability. In the area of work of the intake systems, the following problems can be distinguished:

Available online by August 31st, 2021 https://doi.org/10.15866/irease.v14i4.20223

the boundary layer issues for various types of inlet systems [1], [2] the influence of the jet engine inlet operating conditions on the fan [3], [4] or the inlet vortex problem [5]. In the case of the analysis of the problem of the phenomenon of the inlet vortex itself, as in [5], the mechanism of generating a ground vortex has been analyzed at different variants of the gust (side wind and head wind). The mechanisms of vortex formation for different gust variants are also different. In [5], the authors have created and verified a mathematical model in order to predict the strength of a vortex on the ground based on the aerodynamic similarity. Based on the built model, the effect of ground clearance (inlet-ground) on the vortex strength has been assessed. The problem of a vortex initiated by crosswinds is particularly important, since it can cause a strong disturbance of the flow parameters at the inlet to the engine, which affects its performance [6], [7]. In addition, it should be remembered that the inlet vortex particularly affects the operating range of the compressors and fans themselves, but it should not be forgotten that other factors influence their performance and installation life [8], [9]. In this work, the subject of analysis is the fuselage intake system of a multi-role aircraft propulsion unit. These types of systems are characterized by high susceptibility to intake vortex (Fig. 1). However, the basic task of the intake system is to supply an adequate amount of ambient air to the engine, with a homogeneous field of parameters, with a simultaneous dynamic compression process and as low pressure losses $\left(1-\sigma_{wl}^{*}\right)$ as possible.

An important part of operational design of the intake system is to determine the changes of cross-sectional fields of the intake system, in order to ensure the stability of the entire system (aircraft propulsion). In this area, there are unconventional solutions aimed at improving the operational properties of the engine intake system.

An example of this is the S-duct ridge inlet, which, according to the authors of [2], is to ensure optimal cooperation between the airframe system and the engine.

The authors of the publication have examined the entire structural system of the aircraft in terms of its aerodynamic characteristics, as well as parameters obtained in the inlet flow channel, such as velocity or pressure distributions.

Fig. 1. Aircraft engine intake vortex at F-16 aircraft [10]

The optimal design of the jet engine intake system is to ensure a uniform distribution of the air flow to the engine with minimal pressure losses over a wide range of operating conditions [11]. Such a possibility is provided by the use of the equation of continuity. For example, by analyzing the external section (free) of the airflow (H) and the final section of the intake system (1), assuming that there are no air bleeds, an equal mass flow rate is obtained in relation (1):

$$j\frac{p_H^*}{\sqrt{T_H^*}}A_H q\left(\lambda_H\right)=j\frac{p_1^*}{\sqrt{T_1^*}}A_1 q\left(\lambda_1\right) \tag{1}$$

where $j=f(k,R)$. In the analysis of intake systems, equation (1) transforms frequently into a variable relative value, which is cross-section $\underline{A}$:

$$\underline{A}=\frac{A_H}{A_1}=\sigma_{wl}^{*}\frac{q\left(\lambda_1\right)}{q\left(\lambda_H\right)} \tag{2}$$

Equation (2) applies to freely selected sections of the intake airflow. The relationship shows that in the case of an increase in the design flight Mach number Ma_H, its consequence is an increase in airflow density $q(\lambda_H)$ and dynamic compression. This entails the reduction of the required minimum intake section. However, the relative actual minimum plane should be 5÷15% larger than the theoretical value owing to the problem of the wall layer detachment.

The operation of the intake system is influenced by a number of parameters, which at the same time affect the homogeneity of the distribution of parameter fields at the engine intake, which affects the correctness of engine operation [12]. The nature of changes in the field and parameters of the airflow section are subject to variations depending on, among other things, the flight phase and the operational range of the aircraft engine. The major cases from the stable intake operation perspective include:

- Ground operation;
- Start and take-off;
- Approach to landing;
- Crosswind;
- The range of unstable engine operation.

The operation of the intake system including its operational stability is determined by the following parameters and their changes:

- Intake pressure ratio π_{wl}^{*} ;
- Pressure loss coefficient σ_{wl}^{*} ;
- Flow rate coefficient ϕ_{wl} ;
- Intake drag coefficient $C_{x,wl}$;
- Surge margin ΔK_{wl};
- Distortion coefficient $DC(\Theta)$.

The intake vortex is an important reason in disturbing the intake (Fig. 2) and compressor unit in actual ground conditions. It leads to changes of parameters in the peripheral as well as radial direction.

Fig. 2. Simulation of the intake system of fuselage turbine engine on the ground at a laboratory stand with visualization of the pathline [Photo A. Kozakiewicz]

The consequence of this phenomenon is the impact on the position of the surge line. This leads to a lowering of the distorted surge line of the compressor [13]. In each case, when the flow reaching the compressor or fan shows large heterogeneity of parameters, it has a detrimental effect on the efficiency of the engine [11].

A distorted field of parameters as temperature, pressure, and speed accompanies the airflow through the turbine engine intake system. The disturbance of the parameter field applies to each intake type. The essence of this problem is also strongly visible during changes of engine parameters expressed as a function of time (Fig. 3). The consequence of this type of phenomena may be engine stalling, which is visible for the interval t>190 s [13], as a result of growing deviations of airflow parameters. It is also worth paying attention to the fact that the duration of disturbance of the flow parameters at the engine inlet strongly influences the reaction of the fan or the compressor. The above is due to the finite response time of the compressor or fan. These devices need time to adapt to unstable inlet conditions [15].

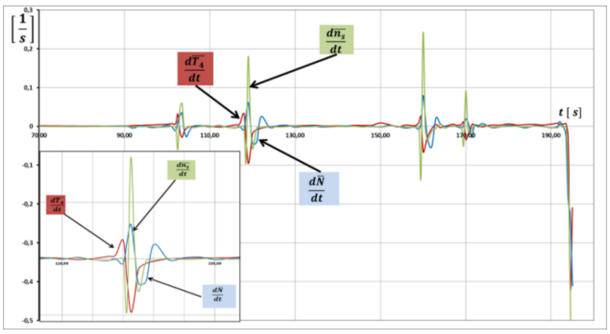

Fig. 3. Characteristics of changes in parameters of turbine engine during surge [14]

The disturbances in the engine inlet flow parameters, caused by a number of aerodynamic aspects, may consequently lead to large distortions of the total pressure flow fields and vortex deformations, leading to a stall on the first stages of the compressor or the fan.

Moreover, discrete distortion areas may have a strong influence on the increase in blade loads, increase in mechanical vibrations and decrease fatigue life [16].

Additionally, disturbing the flow in the form of an inlet vortex may lead to damage to the fan or compressor blades due to the suction of foreign bodies, e.g. from the airport apron [17]. The article aims at presenting the results of numerical analyses concerning the impact of the vortex on distortion of the parameter field in the intake system. This type of analysis allows visualizing the flow phenomena occurring in the inlets of the jet aircraft engines [11], [17] and analyzing their influence on the uneven distribution of the flow parameters along the engine inlet.

The main body of this paper is divided into four sections. Section II presents the design of the computational model, prepared numerical grid and the applied finite volume method. Section III is the most comprehensive section of the article. It contains an analysis of the obtained results. The results are given in the form of raster and bar charts. Particular attention has been paid to the analysis of the influence of the inlet vortex on the unevenness in the pressure distribution along the engine inlet channel. The conclusions of the research and the analyses are presented in Section IV.

II. Design of the computational model

For the purpose of performing numerical flow tests in the F-16 aircraft duct, a numerical model of the front part of its fuselage has been developed (Fig. 4). Then a virtual model has been developed for the numerical simulation process using the following steps:

- Import of a solid object from an external file exported from Siemens NX software;
- Development of the computational area (computational domain);
- Assignment of appropriate boundary conditions;
- Discretization of the model and the computational domain;
- Export of the discretized test object to the Ansys Fluent solver.

Following the import of the model into the Design Modeler software, the development of the computational domain has been launched.

A cuboidal domain of 20 m × 10 m × 10 m has been adopted. Afterwards, from this domain, a smaller computational area has been separated, with the dimensions of 7.16 m × 3.75 m × 4 m, for making a denser numerical grid in the immediate vicinity of the test object. This is a deliberate procedure to ensure greater accuracy of calculations. Fig. 5 shows a developed computational domain with a separate, additional sub-area.

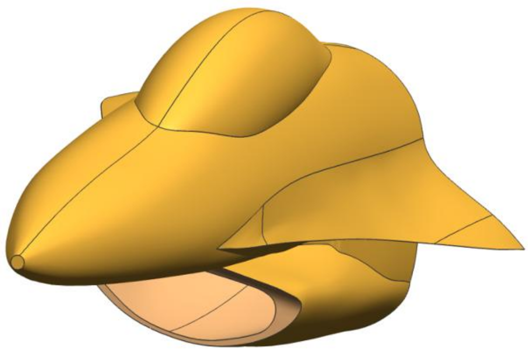

Fig. 4. A model of the front part of the F-16 aircraft fuselage

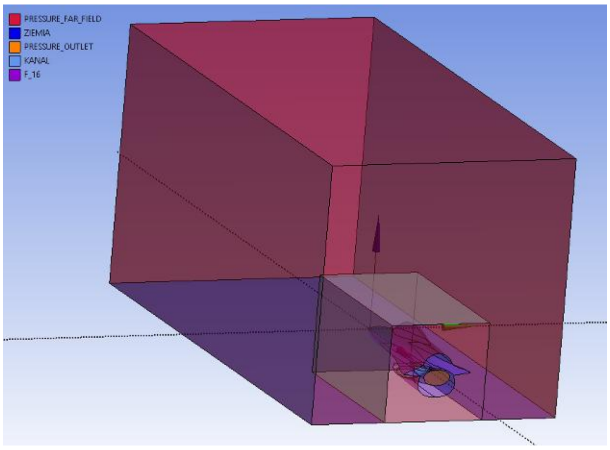

Fig. 5. Names assigned to the planes
of the computational area and the fuselage

Once the computational domain has been developed, efforts have been undertaken to define boundary conditions. To the external surfaces of the computational area, the pressure far field condition has been assigned. It corresponds to the undisturbed flow condition. In the compressor intake plane, a pressure outlet condition has been adopted, which allows changing the vacuum at the outlet from the computational domain, which corresponds to natural operational conditions at the compressor intake. A wall type condition has been applied to the ground and fuselage surfaces. The respective names assigned to the planes are shown in Fig. 5. The next phase has been discretization, which has been carried out using the so-called hybrid grid. It is a combination of structural and non-structural grids. The structural grid has been built in the engine intake duct in the area of the wall layer, while the non-structural grid has been built in the remaining area. In the area of the wall layer, five layers of prismatic elements have been modelled, while the remaining volume of the computational area has been discretized with quadrilateral elements. The total number of grid elements has been 5,681,608, including 325,945 prismatic elements in the boundary layer area. The grid distribution in the plane of symmetry of the front part of the aircraft model is shown in Fig. 6, while the grid distribution in the engine intake area with visible prismatic elements in Fig. 7.

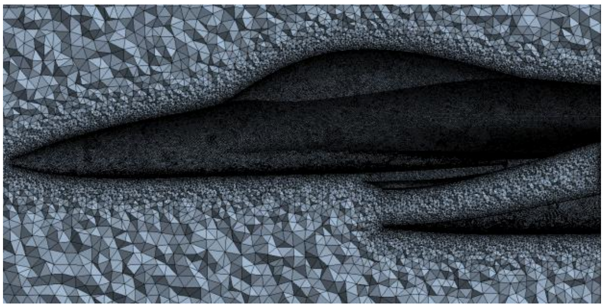

Fig. 6. Grid distribution in the symmetry plane

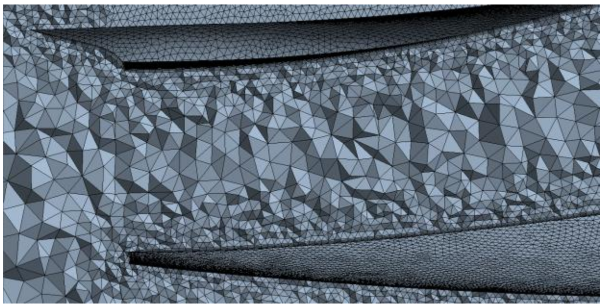

Fig. 7. The grid layout in the engine intake area

The development of the virtual model for the process of numerical simulations has been completed by exporting the discredited test object to the Ansys Fluent computational solver. The commercial package Ansys Fluent CFD software has been used for the calculation.

The package is based on the finite volume method. An indisputable advantage of this method is that it provides for building non-orthogonal and non-uniform computational grids, which is essential for computational tasks involving objects of complex shapes. The method is based on the direct discretization of the equations expressing conservation laws in the physical space.

Therefore, the following conservation equations represent the starting point:

- *Mass conservation equation (continuity equation):*

$$\frac{\partial}{\partial t}\iiint_V \rho dV + \iint_S \rho v_n dS = 0 \tag{3}$$

- *Momentum conservation equation:*

$$\frac{d}{\partial t}\left(\iiint_V \rho \vec{v} dV\right) = \iint_S p_n dS + \iiint_V \rho \vec{F}_m dV \tag{4}$$

- *Energy conservation equation:*

$$\frac{d}{dt}\left[\iiint_V \rho\left(c_V T + \frac{v^2}{2}\right) dV\right] = \iint_S p_n v dS + \\ + \iiint_V \rho \vec{F}_m \vec{v} dV + \iint_S \dot{q}_n dS + \iiint_V \dot{q}_m \rho dV \tag{5}$$

In order to facilitate further transformations, the three equations specified hereinabove can be saved as:

$$\frac{\partial}{\partial t}\iiint_V \vec{\mathbf{\Phi}} dV + \iint_S \vec{\mathbf{H}} ds = \iiint_V \vec{\mathbf{R}} dV \qquad (6)$$

where **Φ**, **H**, **R** are column vectors:

$$\vec{\mathbf{\Phi}} = \begin{bmatrix} \rho \\ \rho\vec{v} \\ \rho e \end{bmatrix} \qquad \vec{\mathbf{H}} = \begin{bmatrix} \rho v_n \\ \rho\vec{v}v_n \\ (\vec{v}\vec{n})\rho e \end{bmatrix}$$

$$\vec{\mathbf{R}} = \begin{bmatrix} 0 \\ \rho\vec{F}_m + div\Pi \\ \rho\vec{F}_m\vec{v} + \vec{q}_m + div(\Pi\vec{v}) + div(\lambda gradT) \end{bmatrix}$$

where Π is the surface stress tensor. A vector **Φ** is a vector of state, and its components include mass, momentum and total energy of a unit of volume. These are the basic values characterizing the physical state of the fluid. The first expression of the left part of the equation (6) determines the speed of change of this state over time, caused by the action of external sources.

External sources also cause the change of momentum and energy. The surface integral in Eq. (6) is a convection element and determines the flows of these quantities through the external surface. The expression on the right is the source element and contains diffusion element under the divergence sign. Then the above equations are averaged in accordance with the formulas referred to in [18], [19] thus obtaining the equations known as Reynolds equations. These equations have been used to address the issue. It should be noted that the effect of the above-mentioned equation averaging operation is that the previously closed system of equations becomes an open system, because six complementary relations, defining components of the turbulent stress tensor, are missing. Hence, the application of turbulence models is needed.

III. Analysis of the Results

The issue of intake system operation analysis is presented in [20]-[24]. The authors of this paper have not encounter such an approach to the problem concerning the impact of the intake vortex on the stability of this type of actual intake. The analysis of the supersonic aircraft's intake system, in a fuselage structural system, has been intended to provide new information about the disturbed distribution of the total pressure field along the duct in case of an existing intake vortex. The problem of intake vortex formation itself has been previously examined by the authors and discussed in [10], [25]-[27].

The case of the intake system with an intake vortex, caused by a crosswind, at an angle of 45°, has been adopted in order to investigate this problem. The selected angle and Mach number of the gust *Ma*=0.0075 has ensured the creation of a vortex in numerical analyses [25]. At the same time, the gust has not stalled the vortex. As a result of the simulations carried out, the pressure variations have been distributed along the duct (Fig. 9). The analysis of pressure changes in the longitudinal section has provided information only about the range of total pressure changes in the intake duct, which has ranged from $p^*=2.01\times10^4$ Pa to $p^*=1.07\times10^5$ Pa.

This analysis does not produce unambiguous data on distorted pressure distribution in the cross sections, which have a significant impact on the operational stability of the compressor. For this purpose, four zones have been adopted for analysis (Fig. 10). They have included the intake section A (4.5 m from the nose of the aircraft fuselage), section D, connecting the intake with the aircraft engine, (6.0 m from the nose) and two intermediate sections have been selected (B - 5.0 m and C - 5.5 m from the nose).

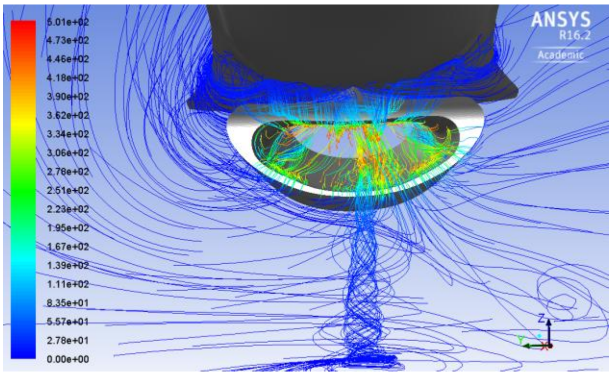

Fig. 8. Front view of current lines

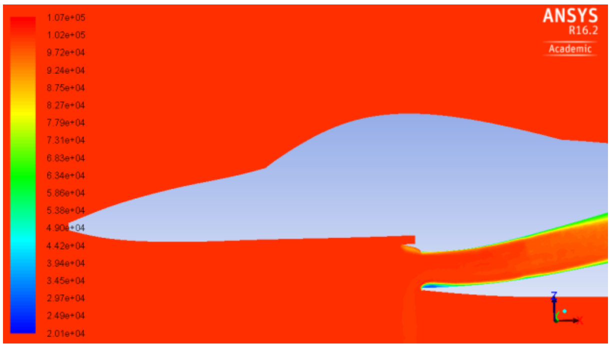

Fig. 9. Total pressure in the symmetry plane

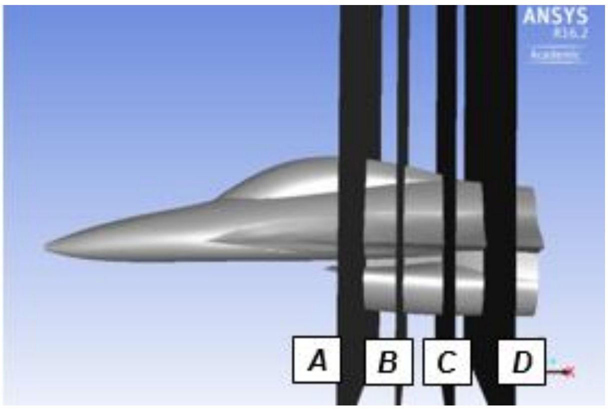

Fig. 10. View of intake system cross-section planes

The obtained results are presented in Figs. 11-14.

Pressure scale is identical with the scale in Fig. 9.

Analyzing the results obtained in the following sections, in terms of quality the following enlargement of the field of disturbance and an increase in the value of the distribution of the impact pressure field have been noticed. The disturbance zones are enlarged from the side parts of the intake duct in parallel to the development of lower zone. The lowest pressure with the value of $p^*=2.01\times10^4$ has been obtained in section A. From these zones, the main areas of reduced impact pressure develop in the flow duct. The lowest can be found in section D and equals 5×10^4 Pa. In addition to the qualitative analysis, a qualitative analysis has been conducted. The pressure area in the sections A, B, C, D has been divided into five compartments:

- Series 1: ($8.75\times10^4 \div 1.07\times10^5$) Pa;
- Series 2: ($7.79\times10^4 \div 8.75\times10^4$) Pa;
- Series 3: ($5.86\times10^4 \div 7.79\times10^4$) Pa;
- Series 4: ($4.42\times10^4 \div 5.86\times10^4$) Pa;
- Series 5: ($2.01\times10^4 \div 4.42\times10^4$) Pa.

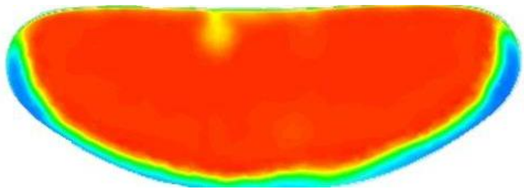

Fig. 11. Total pressure in section A (4.5 m from the nose)

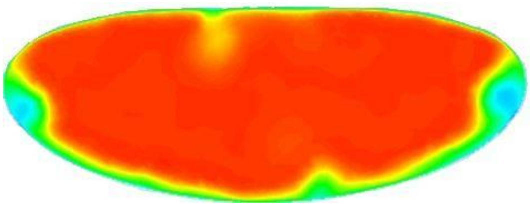

Fig. 12. Total pressure in section B (5.0 m from the nose)

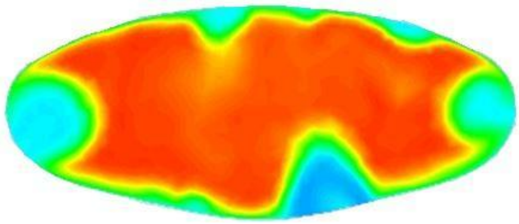

Fig. 13. Total pressure in section C (5.0 m from the nose)

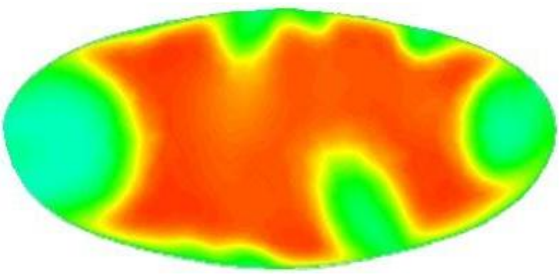

Fig. 14. Total pressure in section D (6.0 m from the nose)

Results are presented in Fig. 15. The graphs show the percentage change of main pressure zones. The percentage share of main pressure zone (Series 1) decreases from 78.9% to 52.1%. There is a zone increase (series) in 2, 3 and 4. With regard to zone 2, the change is from 6.0% to 15.9%, in zone 3 it is from 4.5% to 15.8% and in zone 4 it is from 3.5% to 10.2%. The relative total change of the pressure field outside the main pressure zone (series 1) is shown on Fig. 16. While in section A (4.5 m) and B (5.0 m) the total area of zones (2÷5) is the same, occupying about 17-18% of the area, in zone C (5.5 m), it reaches 34.9%, and in zone D (6.0 m) it is 41.9%. Along the intake duct, the field of disturbed impact pressure increases. Non-dimensional parameters are used for the analysis of uneven distribution of parameters, including total pressure. They represent a combination of a given parameter and surface area, which can be expressed by angular values for axially symmetrical systems [13]. In this paper, since the duct layout is not axially symmetrical, it is proposed to use the surface field in a dimensionless form A_i. The product of pressure p_i and surface area coefficient A_i is introduced, which, in relation to the total value of these figures, gives a dimensionless pressure coefficient (7):

$$\Delta p_{w,i} = \frac{p_i A_i}{\sum_{k=1}^{k=n} p_k A_k} \tag{7}$$

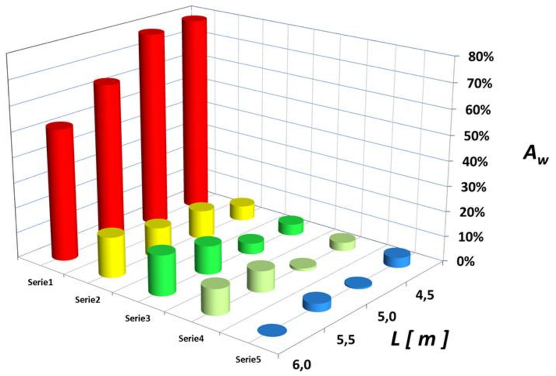

Fig. 15. Analysis of the distribution of changes in pressure fields in the flow duct, where L is the distance from the aircraft nose, A_w is the relative cross-sectional area of a given pressure, total pressures

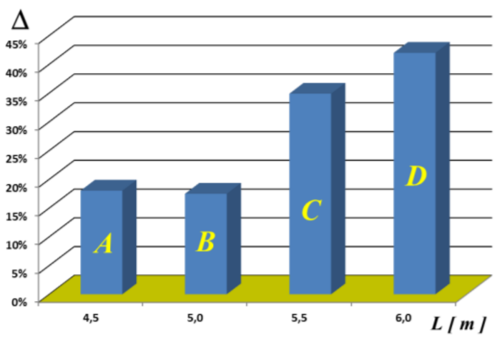

Fig. 16. Analysis of the distribution of changes in pressure fields in the flow duct, where L is the distance from the aircraft nose, Δ is the relative increase of the total pressure area outside the main zone, *A,B,C,D* are the indexes of intake duct cross-sections

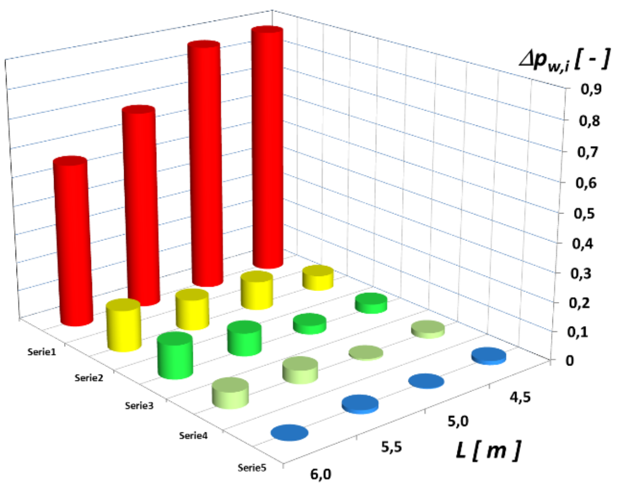

Fig. 17. Analysis of the distribution of pressure coefficient changes in the flow duct, where L is the distance from the aircraft nose, $\Delta p_{w,i}$ is the dimensionless pressure coefficient

The analysis of the pressure distribution on the cross-sectional areas of the fuselage intake duct has provided information about the increasing intensity of the disturbed flow field.

The main field (series 1) declines, which is manifested by the falling value of the coefficient $\Delta p_{w,i}$ from 0.88 to 0.56. This has happened in favour of other zones (series 2 ÷ 4), including in particular series 2 value increase $\Delta p_{w,i}$ from 0.05 to 0.14 and series 3 value increase from 0.03 to 0.11:

$$\Delta p_w(L) = \sum_{i=1}^{i=k} \Delta p_{w,i} \quad (8)$$

For the analysis of the whole section, it has been proposed to use the resultant coefficient presented by the dependence (8).

This total pressure coefficient can be used very well in order to test the homogeneity of the flow field. In the analyzed system, the decrease of this parameter (Fig. 18) illustrates accurately the growing heterogeneity of the parameter field (intensity of changes) in consecutive sections. The highest coefficient value $\Delta p_w(L)$ has been recorded for the B section (L=5.0), which gives the lowest intensity of changes in the airflow field in the intake duct. For C section, its value falls to $\Delta p_w(L)$=0.9362, and for D section to $\Delta p_w(L)$=0.8795. In section B, numerical analyses have showed the lowest intensity of flow disturbances. Therefore, this cross-section has been adopted as a reference cross-section. In the intake duct, heading for the engine, there is an increasing intensity of flow field disturbance, whereas in the C-section it is higher by 7.26% and in D-section by 12.88%. The above analysis of the obtained results provides information about the increasing unevenness of the pressure distribution along the channel. The increasing unevenness in the pressure distribution is dangerous from the engine operating point of view and the possibility of the engine stalling if the compressor has surged [13].

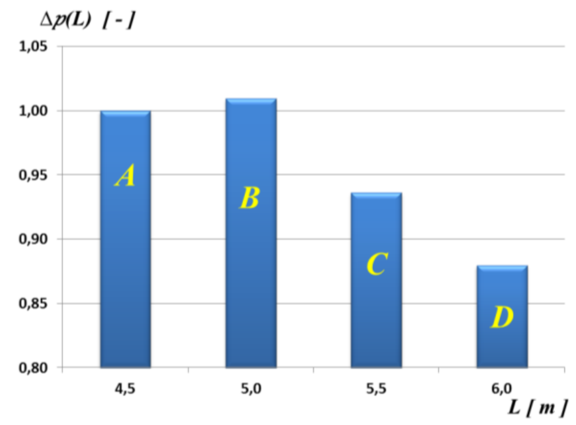

Fig. 18. Distribution of pressure coefficient changes along the engine intake duct, where L is the distance from the aircraft nose, A, B, C, D are the indexes of intake duct cross-sections

IV. Conclusion

The analysis has provided interesting results on the impact of the vortex on the stability of the intake system, which has been the objective of this paper. The study has focused on the problems of vortex and a risk of sucking in foreign objects (FOD) from the airfield surface. The results obtained and the introduced coefficients have provided information about the growing disturbance along the duct, which is dangerous from the engine operation perspective and a risk of engine stalling, if the compressor has surged. This information is also important for the programming of a control system that should protect the engine during the control process during operation in transient ranges. It can be considered that the proposed dimensionless pressure coefficient meets the expectations for the assessment of airflow heterogeneity. The research team is going to use it in the follow-up analyses concerning the analysis of the flow field in the intake system under evolving conditions of vortex formation, due to the importance of this issue in terms of operational safety of the turbine jet engine.

Acknowledgements

This work was financed by Military University of Technology under research project UGB 898/2021.

References

[1] I. Arif, S. Salamat, M. Ahmed, F. Qureshi and S. Shah, Comparative Flow Field Analysis of Boundary Layer, Diverter Intake and Diverterless Supersonic Intake Configuration, *Journal of Applied Fluid Mechanics*, Vol. 11, No. 4, pp. 1125-1131, 2018

[2] Bravo-Mosquera P. D., MartinsAbdalla A., DaríoCerón-Muñoz H., Catalano F. M., Integration assessment of conceptual design and intake aerodynamics of a non-conventional air-to-ground fighter aircraft, *Aerospace Science and Technology* 86, p. 497–519, 2019.
doi: https://doi.org/10.1016/j.ast.2019.01.059

[3] Gil-Prietoa D., MacManusa D. G., Zachosa P. K., Bautistaa A., Assessment methods for unsteady flow distortion in aero-engine intakes, *Aerospace Science and Technology*, Volume 72, p. 292-304, January 2018.

doi:10.1016/j.ast.2017.10.029
[4] H. A.J. G. Beik, S. H. Torabi, and H. B. Tabrizi, Stall Margin Improvement and Increase Pressure Ratio in Transonic Axial Compressor Using Circumferential Groove Casing Treatment, *AUT J. Mech. Eng.*, vol. 3, no. 1, pp. 107–112, 2019
[5] Jia W., Wu Y., Lei Y., Generation mechanism and aerodynamic characteristic modeling of ground vortex in crosswind condition, *Aerospace Science and Technology*, 99 10558, p.1-9, 2020. doi: https://doi.org/10.1016/j.ast.2019.105581
[6] Jia, Y., Liang, H., He, Q. et al. Flow Separation Control of Nacelle in Crosswind by Microsecond Pulsed Surface Dielectric Barrier Discharge Plasma Actuator, *Flow Turbulence Combust*, 2021.
[7] I Liu H., Yi B., Wang C., Hui M.: Numerical simulations on nacelle inlet ground vortex under crosswind conditions, *Aeroengine*; Nanjing, China, 43(6), 2017
[8] Hassan, H., Gobran, M., El-Saied, A., Turbofan Engine Performance Deterioration Due To Fan Erosion, (2019) *International Review of Aerospace Engineering (IREASE)*, 12 (2), pp. 93-100. doi: https://doi.org/10.15866/irease.v12i2.16072
[9] Hassan, H., Gobran, M., El-Azim, A., Performance Prediction for the Fan of the CF6-50 Turbofan Engine at the Off-Design Conditions, (2013) *International Review of Aerospace Engineering (IREASE)*, 6 (1), pp. 9-17.
[10] Kozakiewicz A., Frant M., The analysis of the gust impact on inlet vortex formation of fuselage-shielded inlet of a jet engine powered aircraft, English, *Journal of Theoretical and Applied Mechanics JTAM*, vol.51, no. 4, pp. 993-1002, Warsaw, Poland. 2013.
[11] Yeung A, Vadlamani NR, Hynes T, Sarvankar S. Quasi 3D Nacelle Design to Simulate Crosswind Flows: Merits and Challenges. *International Journal of Turbomachinery, Propulsion and Power*, 4(3):25, 2019.
[12] W. Zhang, S. Stapelfeldt, and M. Vahdati, Influence of the inlet distortion on fan stall margin at different rotational speeds, *Aerosp. Sci. Technol.*, 2020. doi: 10.1016/j.ast.2019.105668
[13] Seddon J., Goldsmith E.L., *Intake Aerodynamics*, AIAA Education Series, 2nd ed., Virgina 1999.
[14] Kozakiewicz A., *Estimation of the operating point in order to optimize the geometry of the stockade compressor*, Poznan University of Technology Publisher, Poznań 2013, pages: 138.
[15] D. Gil-Prieto, D. G. MacManus, P. K. Zachos, A. Bautista, Assessment methods for unsteady flow distortion in aero-engine intakes, *Aerospace Science and Technology*, Volume 72, pp. 292-304, 2018.
[16] Tanguy, D. G. MacManus, E. Garnier, P. G. Martin, Characteristics of unsteady total pressure distortion for a complex aero-engine intake duct, *Aerospace Science and Technology*, Volume 78, pp. 297-311, 2018.
[17] Millot G., Raynal S., Laurencin A., *Benefit of mesh adaptation vs. conventional CFD approach for nacelle aerodynamics with ground effect*, Conference: NAFEMS France 2020.
[18] Lumley J.L., Yaglom A.M.: *A Century of Turbulence*, Proc. VII European Turbulence Conference, Barcelona, 2000.
[19] Wilcox D.C.: *Turbulence modeling for CFD*, 2000.
[20] Anderson B. H., Study on Vortex Flow Control of Inlet Distortion, *journal of propulsion and power*, Vol. 8, No. 6, Nov.-Dec. 1992.
[21] Arif I., Salamat S., Ahmed M., Qureshi F., Shah S., Comparative Flow Field Analysis of Boundary Layer Diverter Intake and Diverterless Supersonic Intake Configuration, *Journal of Applied Fluid Mechanics*, Vol. 11, No. 4, pp. 1125-1131, 2018.
[22] Ibrahim I.H., Ng E.Y.K., Wong K., Flight Maneuverability Characteristics of the F-16 CFD and Correlation with its Intake Total Pressure Recovery and Distortion, *Engineering Applications of Computational Fluid Mechanics*, Vol. 5, No. 2, pp. 223–234, 2011.
[23] Karlsson A., Fuchs L., Vortex systems and the interaction between an air inlet and the ground, *ICAS 2000 CONGRESS*, ICAO522, pp.522.1-522.10
[24] Lotter K.W., Mackrodt P. A., *Engine surge simulation in wind-tunel model inlet ducts*, ICAS-88-4.11.4, *ICAS and AIAA*, pp. 1773-1788, 1988.
[25] Kozakiewicz A., Development of a Model for the Intake Channel within a MRCA Turbine Engine in Order to Analyze Intake Vortex Phenomena, *the Solid State Phenomena* vols. 198 and 199, pp.188-193, Zurich, 2012, Switzerland.
[26] Kozakiewicz A., Frant M., Numeric analysis of the intake vortex formation in the case of a double fuselage shielded inlet, *Journal of Theoretical and Applied Mechanics*, 52, 3, pp. 757-766, 2014, Warsaw, Poland.
[27] Kozakiewicz A., Frant M., Kachel S., Analysis of impact of gust angle and velocity on the position of stagnation point, *Advances in Science and Technology research*, ASTRJ-01362-2020-02,14, 4, pp.49-57, 2020, Poland

Authors' information

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

Prof. **Adam Kozakiewicz** PhD. Dsc. Eng. earned his M.Sc. in Technical Aircrafts Maintenance at the Military University of Technology – MUT (1987) in Warsaw, Poland, and his Ph.D. in Aerospace Engineering at the Military University of Technology (1995) in Warsaw, Poland. Doctor of Science since 2014. He has been with Military University of Technology for 32 years. With more than 30 year's experience in Theory of Turbine Engines and in Design, Exploitation and Diagnostics of Turbine Engines. Now Professor at Mechatronics, Armament and Aerospace Faculty including lectures on Aircraft and Rockets Engines. The author of 100 articles from the field of air issues. The co-author of five monographs, including the two-volume-monograph „Aircraft engines. Construction-Exploitation-Diagnostics"(part 1 – pp. 428 – Warsaw 2010, part 2 – pp. 460 - Warsaw 2012).

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 more then 20 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 24 articles and one monograph.

Maciej Majcher (Corresponding Author), M. 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. He is carrying out a doctorate in the field of three-dimensional flow analysis in key elements of axial fans. He is a co-author of 9 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

