The Influence of the Level of the Flow Path Blockage at the Inlet on the Fan Characteristics

Grigorii Popov, Oleg Baturin, Andrei Volkov, Daria Kolmakova, Vasilii Zubanov,
Anastasia Korneeva and Yulia Novikova
Samara National Research University, Samara, Russia

Keywords: Nonuniformity, Fan, Turbofan Engine, Low Pressure Spool.

Abstract: The paper presents the results of numerical simulation of the effect of flow nonuniformities at the engine inlet on the working process of the engine fan. Flow nonuniformities is created by pushing the interceptor into the flow part of inlet device like as it is often done during field tests. The authors have created a numerical model capable of considering non-stationary processes in the fan using nonlinear harmonic analysis. As a result, qualitative and quantitative estimates were obtained of the influence of overlapping of the inlet duct by the interceptor on the main parameters of the fan workflow. It is shown that the more the duct is blocked, the more its parameters are deteriorated. Moreover, the deterioration is not linear, but according to the dependence of the 2nd order.

NOMENCLATURE

G mass flow rate of the working fluid, kg/s;
p* total pressure, Pa;
T* total temperature, K;
n rotor speed, %;
m bypass ratio;
η efficiency;
Y+ non-dimensional wall distance;
α flow angle, degree;
LPC low pressure compressor;
RW rotor wheel;
GV guide vane;
NLH nonlinear harmonic analysis.

Note. The flow angles in this research are measured from the aerofoil cascade front.

1 INTRODUCTION

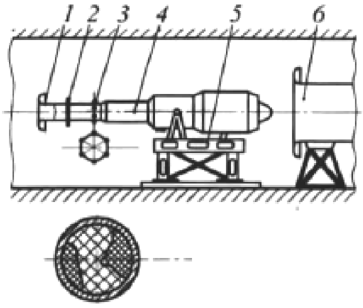
Inlet devices of bypass turbofan engines (characterized by a high degree of bypass ratio) of modern passenger aircraft have a large flow area and are relatively short (their length is less than the diameter). It would seem that losses in such conditions should be minimal in all typical flight conditions. However, in some cases (for example, with a strong side wind, flying sideways, etc.) a separation flow occurs at the inlet edge of the air intake, which causes the flow at the fan inlet to become uneven. This, in turn, causes a significant reduction in the efficiency of the low-pressure compressor and the engine. In addition, the inlet nonuniformity causes oscillations of the fan blades, which can lead to their destruction.

For a long time, the study of the influence of inlet nonuniformity on the GTE workflow was carried out during field tests with an aircraft air intake or its simulator (Figure 1). The latter is a complex of resistances (grids, plates, struts), located between the lemniscate attachment and the engine, creating the same uneven velocity field at the fan inlet, as the aircraft air intake on the flight mode of interest (Grigor'ev, 2009).

At present, in connection with the development of numerical methods for modelling gas-dynamic processes and strength calculations, it has become possible to model the influence of the inlet nonuniformity on the workflow of the fan and the engine. This will allow an assessment of the influence of nonuniformity at the stage of calculations, without manufacturing many prototypes. As a result, at the initial design stage, the design variants that do not work satisfactorily under the specified conditions will be eliminated, which will significantly reduce the time and cost of engine development.

In this paper, the authors aim to test the possibility

Popov, G., Baturin, O., Volkov, A., Kolmakova, D., Zubanov, V., Korneeva, A. and Novikova, Y.
The Influence of the Level of the Flow Path Blockage at the Inlet on the Fan Characteristics.
DOI: 10.5220/0007836502470254
In *Proceedings of the 9th International Conference on Simulation and Modeling Methodologies, Technologies and Applications (SIMULTECH 2019)*, pages 247-254
ISBN: 978-989-758-381-0



1 - lemniscate tip; 2 - simulator; 3 - instruments controlling the flow irregularity; 4 - the engine; 5 - thrust measuring device; 6 - ejector tube

Figure 1: Engine test system with an inlet nonuniformity simulator (Grigor'ev, 2009).

of conducting a computational study of the influence of the inlet nonuniformity on the working process (efficiency) of a fan of turbofan engine, reproducing tests with an input simulator. There will also be given a qualitative and quantitative assessment of the influence of the inlet nonuniformity on the main parameters of the working process of the engine fan.

2 TEST OBJECT

The object of the study was the fan of the NK-56 turbojet engine, developed at Kuznetsov, PJSC (Samara, Russia) (JSC "Kuznetsov", 2019) for civil aviation aircraft in the early 1980s, but not commercialized. The main parameters of the NK-56 engine are given in Table 1 (Zrelov, 2002).

The appearance of the investigated fan is presented in Figure 3. The number of blades is 30. Information about the geometry of the fan and some of its test results (in the form of internal reports of the company) was transferred by Kuznetsov, PJSC to Samara National Research University (Samara University, 2019) s part of joint research.

Table 1: The main parameters of the NK-56 engine (Zrelov, 2002).

Thrust, kN	177
Pressure ratio	25.5
Gas temperature before the turbine, K	1571
Bypass ratio	4.8
Specific fuel consumption (M = 0, H = 0), kg/N h	39.1
Specific fuel consumption (M = 0.8, N = 11 km), kg/kN h	63.75
Outer diameter, m	2.05
Weight with reverse, kg	3340

To conduct the research, a computational model of the LPC of the NK-56 engine was created using the Numeca FineTurbo software (NUMECA, 2008), which includes a fan with an add stage.

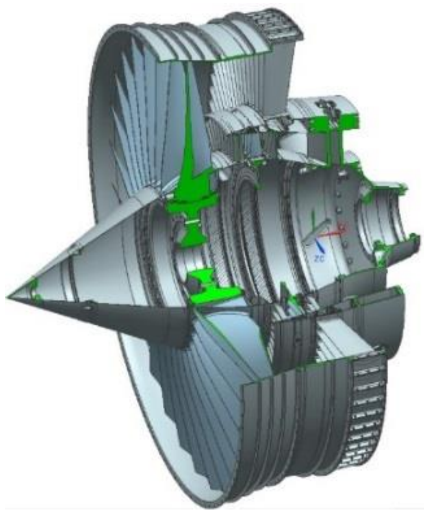


Figure 2: Fan of the NK-56 engine.

Spalart-Allmaras and k-epsilon turbulence models were used in the calculations.

The geometry of the computational domain was created in accordance with the drawings submitted by PJSC Kuznetsov. The computational area included the inlet section, the rotor (RW) and the fan guide vanes (GV), the bypass section and the second stage of the LPC with the engine annular frame (Figure 3). In constructing the model, the deformation of the working blades from the forces acting on them was considered. For this purpose, a preliminary calculation of the fan workflow was carried out at a rotor speed of 100%. Obtained gas loads acting on the blades, were transferred to the ANSYS Mechanical. It identified the deformation of the aerofoil, arising under the action of centrifugal load and gas forces. Then, information on the geometry of the deformed aerofoil was transferred to the Numeca FineTurbo to clarify the gas loads. In total, four such iterations were performed. The criterion for the convergence of iterations was the absence of a change in more than 1% of the deformed shape of the blade aerofoil during the subsequent iteration.

The appearance of the resulting computational model is shown in Figure 3.

The values of the total pressure p^*=101325 Pa and the total temperature T^*=288.15 K were set as the boundary conditions at the inlet to the computational domain. The flow rate of the working fluid was set at the outlet from each circuit. The ratio of mass flow ratios at the outlet from each circuit advised the

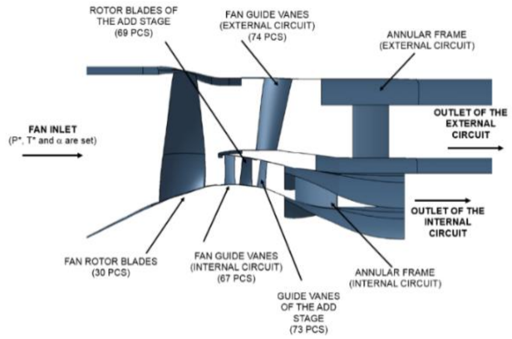


Figure 3: Appearance of the design model of the fan with add stages.

required bypass ratio (for the rotor speed n = 95% - m = 4.9; for n = 100% - m = 4.8; for n = 105% - m = 4.7).

Two mesh models were created for the computational model: light (in total, 2.36 million final volumes - Y+ is more than 7) and heavy (7.06 million final volumes, Y + is more than 2).

4 VERIFICATION OF THE COMPUTATIONAL MODEL

At the first stage of the study, the adequacy assessment and validation of the created computational model was carried out. For this, the calculated characteristics obtained using various stationary models differing in the turbulence model and density of the finite volume mesh were compared with the test results on the engine test bench provided by Kuznetsov, PJSC. Due to the large time that has passed since the tests, a detailed description of the experimental setup and the error estimates of the test data were not provided.

A total of 4 different computational models were created:

No. 1 — *Spalart-Allmaras (SA)* turbulence model, the number of final volumes is 2.36 million;

No. 2 — *k-epsilon* turbulence model, the number of final volumes is 2.36 million;

No. 3 — *Spalart-Allmaras (SA)* turbulence model, the number of final volumes is 7.06 million;

No. 4 — *k-epsilon* model of turbulence, the number of final volumes is 7.06 million.

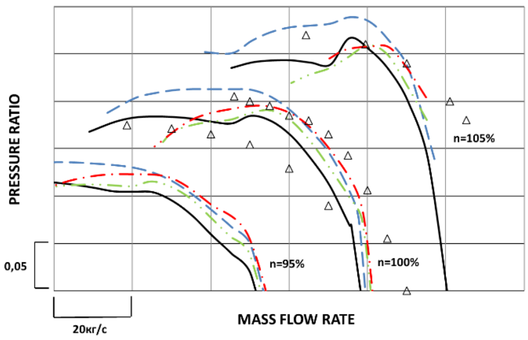
Comparison of the results obtained as a result of stationary calculation with experimental data is shown in Figures 4 and 5 and in Table 2.

Table 2: Comparison of the results of calculations obtained using the considered numerical models with the data by PJSC "Kuznetsov".

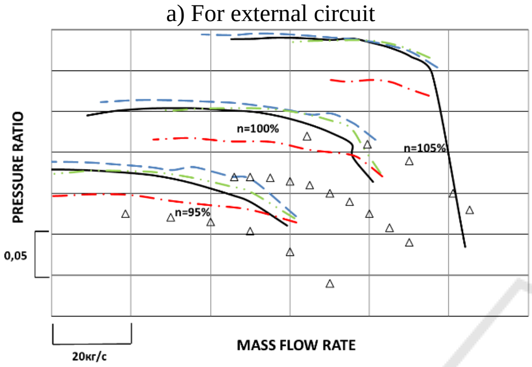
The deviation of calculations by the numerical model relatively data by PJSC "Kuznetsov"	Model No.			
	1	2	3	4
	mesh 2.36 mln.		mesh 7.06 mln.	
	SA	k-ε	SA	k-ε
Internal circuit efficiency	0.5...2% more	0...1% more	1.5% more	1...3% more
External circuit efficiency	3...6% more	3...6% more	2% more	1.5...5 % more
Mass flow rate of the internal circuit	6% less	4% less	±2%	3% less
Mass flow rate of the external circuit	±2%	±2%	±2%	±2%
π_c^* of the internal circuit	Higher by 0.05	↑ by 0.03	Coincides	↑ by 0.02
π_c^* of the external circuit	↑ by 0.07... 0.13	↑ by 0.05... 0.12	↑ by 0.02	↑ by 0.03... 0.1

Analysing the data presented in Figures 4...5 and Table 2, we can come to the following conclusions:

- none of the created computational models show complete agreement with the experimental characteristics in the whole considered range of parameters;
- all computational models show significantly overestimated values of the pressure ratio in the external circuit, but at the same time, they well predict the characteristics of the internal circuit;
- all computational models show overestimated efficiency values, especially in the external circuit (the difference reaches 6%);
- the value of the working fluid mass flow rate, at which the maximum efficiency is achieved, for the external circuit is in good agreement with the data of the design calculation, while for the internal circuit, the resulting flow rate is usually underestimated by 2...4%;
- the smallest discrepancy for the external circuit is observed at a high rotor speed (n = 105%), and for the internal circuit - at the small (n = 95%);
- considering the real deformation of the blade with the help of coupled simulation of strength and gas dynamics allows to reduce the quantitative discrepancy between the calculation data and design data. Qualitatively, the nature of the calculated characteristics of the LPT does not change.



a) For external circuit



b) For internal circuit

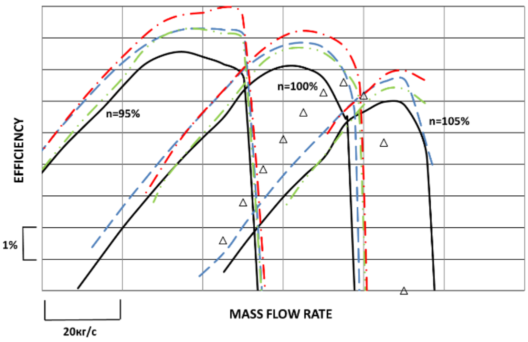
△ - experimental data; — — - model No.1;
—— - model No.2; — . — - - model No.3;
— . . - model No.4

Figure 4: Comparison of the pressure characteristics obtained using the created computational models with experimental data.

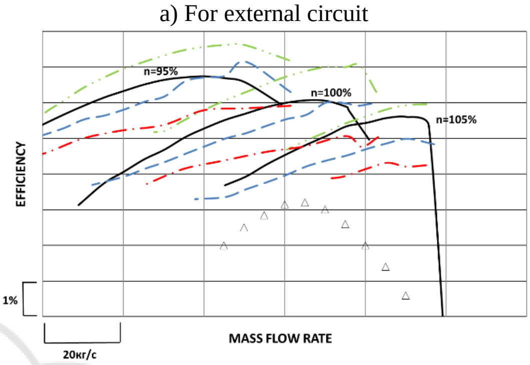
Of all the models considered, the best match with the data of the design calculation is shown by model No. 3 (fine mesh and Spalart-Allmaras turbulence model). It is accepted as the final for further research of the workflow in the fan blade passages. The Mach number contours obtained using this model are shown in Figure 6.

5 UNSTEAY (NLH) COMPUTATIONAL MODEL

At the second stage of the study, based on the created and verified computational model of the workflow of the LPC of the NK-56 engine, a computational model was created to study the influence of the inlet nonuniformity on the working process of its fan.
The modified model was created in such a way as to meet the conditions of testing a fan with a simulator of the inlet nonuniformity on the test benches of PJSC Kuznetsov. There, the nonuniformity is modelled by extending the interceptor into the duct between the lemniscate and the inlet to the fan (Figure 7). During



a) For external circuit



b) For internal circuit

△ - experimental data; — — - model No.1;
—— - model No.2; — . — - - model No.3;
— . . - model No.4

Figure 5: Comparison of the efficiency characteristics obtained using the created computational models with experimental data.

the tests, the intensity of the inlet nonuniformity was regulated by the depth of the interceptor extension to the flow part of the duct.

Due to the highly variable nature of the flow at the fan inlet, the task should be solved in a transient statement. In this case, since the nonuniformity generated by the interceptor has a different intensity of direction of the velocity vectors around the circumference of the RW, the assumption about the periodicity of the flow cannot be accepted, and the computational model must contain all the blade passages.

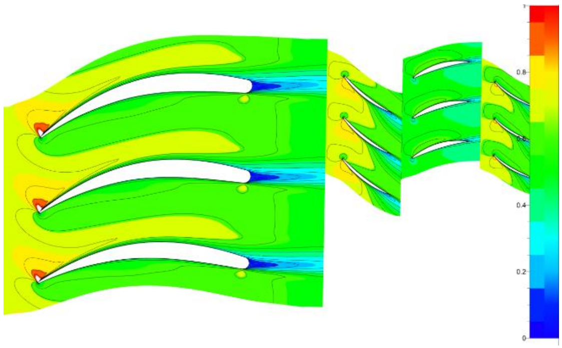
Due to the highly variable nature of the flow at the fan inlet, the task should be solved in a transient statement. In this case, since the nonuniformity generated by the interceptor has a different intensity of direction of the velocity vectors around the circumference of the RW, the assumption about the periodicity of the flow cannot be accepted, and the computational model must contain all the blade passages.

The solution to the problem of studying the influence of inlet nonuniformity on the workflow of the LPC using a transient simulation of a full circle model (containing all passages of LPC of the turbofan engine) requires exorbitant computer resources, and cannot be successfully done in a reasonable time using available computer equipment. For this reason, it was assumed to conduct the research using the method of nonlinear harmonic analysis (NLH) (Vilmin et al., 2013), which allows to obtain nonstationary flow patterns several times faster than using transient simulation. The method allows to obtain non-stationary flow fields by means of decomposition of periodic oscillations in Fourier based on a preselected number of harmonics, usually associated with the transmission frequencies of the blades of the turbomachine configuration and their multiples. At the same time, only one blade passage is required for analysis. This approach allows to obtain pictures of dynamic processes by 2 orders of magnitude faster than with transient calculation (NUMECA International The Nonlinear Harmonic module, 2019).

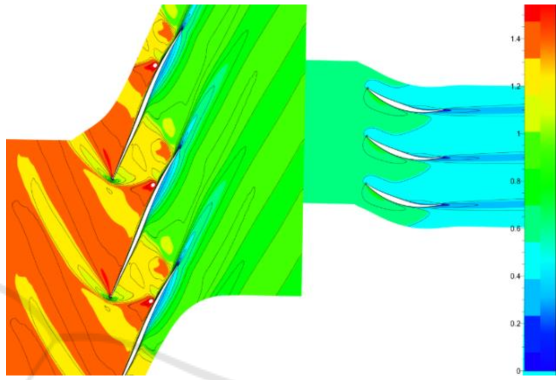
The number of harmonics used in nonlinear harmonic analysis is 3. A series of calculations was also carried out with the number of harmonics equal to 7. The results obtained differed little from the data obtained with 3 harmonics, but in the case of 7 harmonics the solution process was significantly less stable.

The NLH method could not be applied to the required number of blade rows and when the compressor was operating simultaneously for 2 circuits. For this reason, the geometry of the LPC was significantly simplified: the separator of the contours was eliminated (the task became single-circuit). The computational domain contained only a RW and a model GV ("lengthened up" GV of the fan of the internal circuit) (Figure 7) simulating the effect of downstream elements on the rotor.

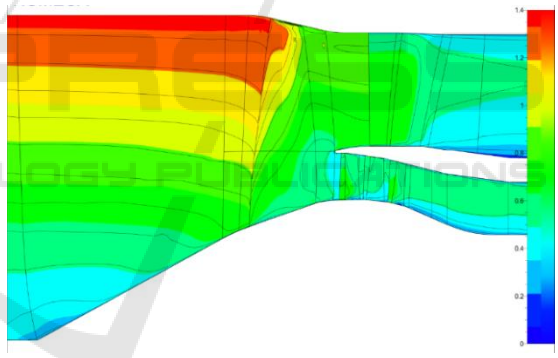
An input section imitating the engine inlet channel with an interceptor was attached to the inlet boundary of the fan domain (Figure 8). Its geometry was created in the Numeca IGG software. Several variants of its geometry were created, differing in the length by which the interceptor was extended. The total number of finite volumes of the computational model shown in Figure 9 is 4 mln. The mesh models of the RW and GV domains were made with the settings corresponding to the Model No. 3 of the LPC (see above).



Hub section of the fan



Shroud section of the fan



The averaged values in the meridional section

Figure 6: Contours of Mach numbers in relative motion in the fan at the operating point at n = 100%.

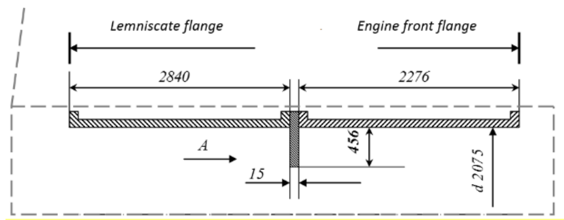


Figure 7: Geometry of the simulated variant of the interceptor in the inlet device when testing the NK-56 engine.

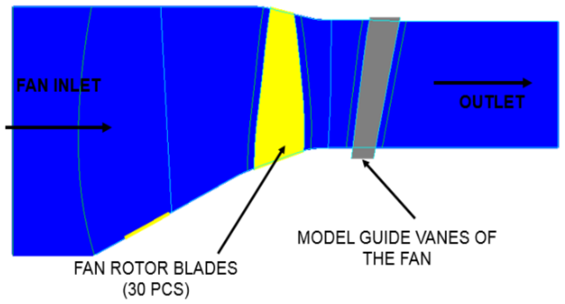


Figure 8: Simplified geometry of the NK-56 engine fan for conducting research on the effect of inlet nonuniformity on its workflow.

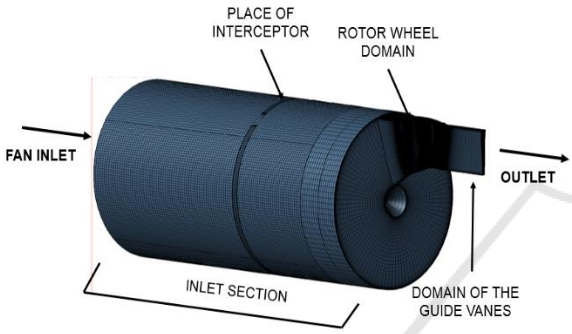


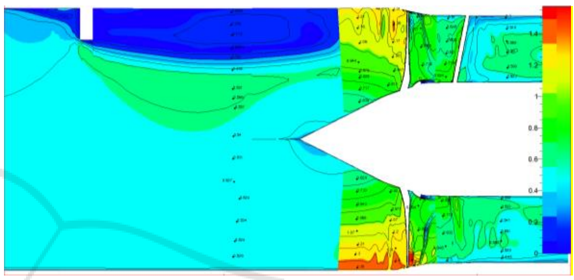
Figure 9: Computational model for studying the influence of inlet nonuniformity on the fan workflow.

6 DISCUSSION OF THE RESULTS

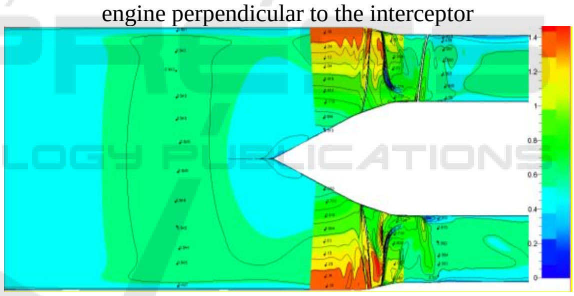
Figure 10 shows the contours of Mach numbers in relative motion when the interceptor extends into the flow part so that it covers 6.8% of the flow part of the duct, in two mutually perpendicular planes passing through the engine axis. It shows that in front of the interceptor a zone of flow deceleration is formed, and behind it a developed separation zone reaches the inlet of the engine, located at a distance more than the size of its outer diameter (i.e. by more than one calibre). In this case, part of the flow entering the duct opposite the interceptor is redirected to the axis of the engine, causing local flow acceleration, and changing the flow structure there. That is, the injecting the interceptor affects not only the structure immediately near it, but the rest of the duct to a depth of more than half the diameter of the engine.

These circumstances lead to the emergence of significant inhomogeneity over the cross section of the pressure field at the fan inlet (Figure 11). It can be seen that it is formed behind the interceptor, and, with an increase of the interceptor extension, an area of reduced total pressure grows behind it. In this case, Figure 11 confirms that with an increase in the overlapping area of the duct, the uniformity of the pressure field is disturbed over the entire cross section. And the higher the extension, the greater the level of unevenness.

The result of a quantitative assessment of the effect of the interceptor extension level on the nonuniformity of the pressure field is shown in Figure 12. There, the criterion of non-uniformity applies a value equal to the ratio of the minimum total pressure in the cross section to its maximum value. The smaller this value, the greater the uneven flow. As can be seen from Figure 12, the flow nonuniformity with increasing overlap of the flow path by the interceptor increases linearly. Moreover, when 13% overlap, the nonuniformity reaches 50%.



a) Section plane No.1 passing through the axis of the engine perpendicular to the interceptor

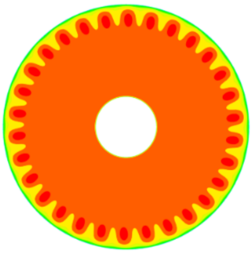


b) Section plane No.2 passing through the axis of the engine perpendicular to plane No. 1

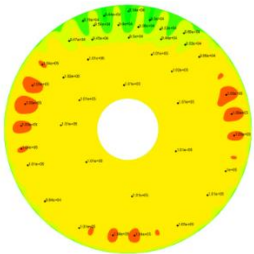
Figure 10: The calculated contours of the change in the Mach number in relative motion in the “inlet duct+ fan” system when the interceptor is extended so that it blocks 6.8% of the flow-part of the duct.

The above-described phenomena lead to the fact that the conditions at the inlet to each blade passage and, correspondingly, the flow structure there are unique (Figure 13), disrupting the interaction of adjacent passages and reducing the pressure ratio, efficiency, stability margins and air flow through the fan.

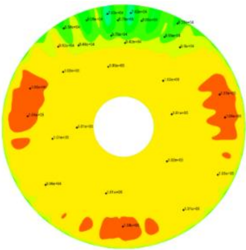
The effect of nonuniformity on the fan workflow parameters at a rotation frequency of n = 100% can be estimated from the evolution of the characteristics as the inlet device is overlapped by the interceptor (Figure 14). It can be seen from the above data that



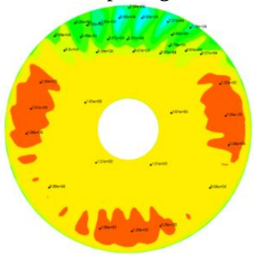
a) No interceptor



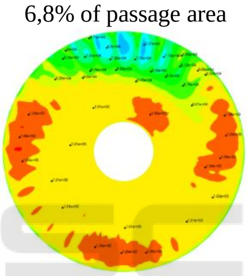
b) Interceptor overlaps 4,9% of passage area



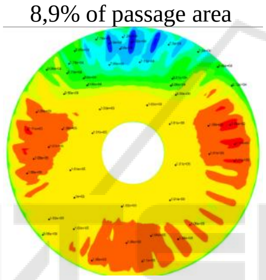
c) Interceptor overlaps 6,8% of passage area



d) Interceptor overlaps 8,9% of passage area



e) Interceptor overlaps 11.2% of passage area



f) Interceptor overlaps 13,5% of passage area

Figure 11: Transformation of the total pressure fields at the fan inlet flange at different levels of extension of the interceptor (at the top) to the flow part of the supply duct.

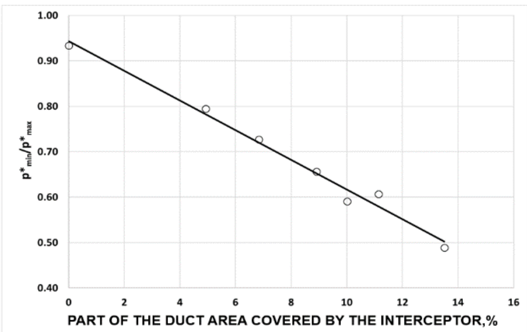


Figure 12: Influence of the overlap level of the input duct by the interceptor on the unevenness of the total pressure field at the fan inlet.

with the increase in the part of the duct blocked by the interceptor, all compressor parameters deteriorate: the working fluid mass flow rate, the pressure ratio, and the efficiency decrease. In addition, the range of mass flow rate between the modes of surge and choke is also reduced, pressure lines become more vertical. This signals a decrease in the stability of the compressor.

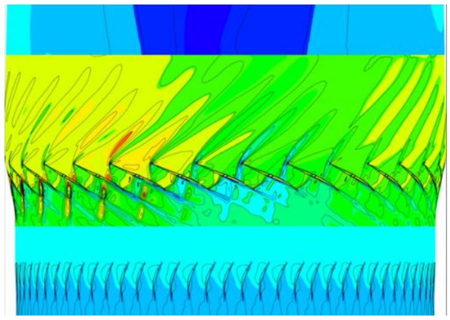
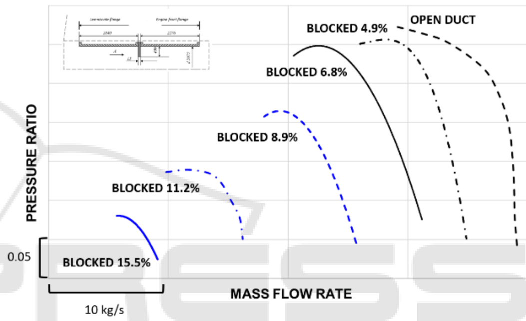
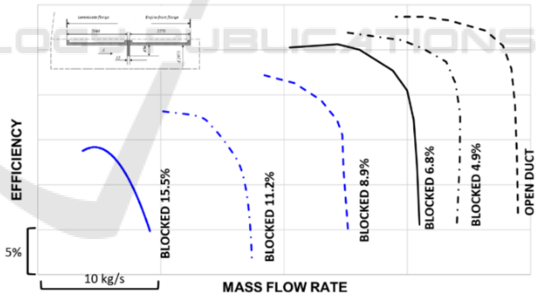


Figure 13: Contours of Mach numbers in relative motion in the peripheral part of the fan (height of 98%) opposite the installation site of the interceptor.



Pressure characteristics



Efficiency characteristics

Figure 14: Comparison of fan characteristics at n = 100% at different values of the interceptor extension into the flow part.

A quantitative assessment of the influence of the overlap of the flow part of the input duct on the parameters of the compressor workflow, obtained from the analysis of Figure 14, is shown in Figure 15. As can be seen, the increase in the level of the interceptor extension degrades the parameters not linearly, but by parabolic dependence. The least change is in the mass flow rate of the working fluid (with a decrease in the area of the inlet duct by 10%, it is reduced by 3%). The difference between the values of mass flow rate at surge and choke changes most of all (with a decrease in the duct area by 10%, it is reduced by 40%). If the duct area is reduced by 10%, the efficiency of the compressor decreases by 11% (rel.), and the pressure ratio - by 9%.

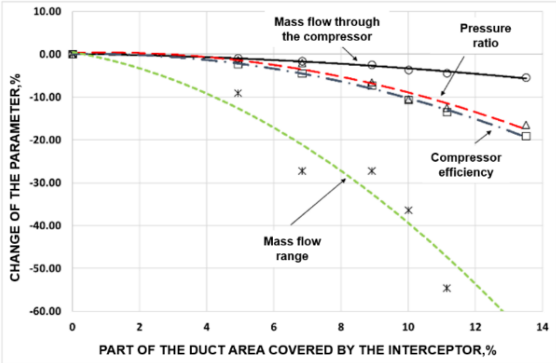


Figure 15: Changing the main parameters of the compressor process at different levels of overlapping the inlet section.

7 CONCLUSIONS

As a result of the performed work, the complex scientific and technical problem of a reliable computational study of the influence of the inlet nonuniformity on the working process of a fan of a turbojet bypass engine was solved. The solution to this problem is hampered by the fact that the process under study is transient and leads to the fact that different fan blade passages operate in different conditions. Such a problem should be solved in a transient setting using a full circle model. However, such a computational model, in addition to the enormous calculation time inherent in transient problems, contains the number of finite elements in the hundreds of millions, which, due to the limited possibilities of the computer equipment, is unacceptable for the authors of the paper.

To solve this problem, the authors developed and verified a numerical computational model of the workflow of an inlet device, retractable interceptor (an inlet nonuniformity generator) and a turbofan fan stage using NLH approach.

In the future, the results are planned to be transferred to the module of the strength calculation and to evaluate the effect of inlet nonuniformity on the static and dynamic loads on the fan blades.

ACKNOWLEDGEMENTS

This work was supported by the Russian Federation President's grant (project code MK-3168.2019.8).

REFERENCES

Grigor'ev, V.A., 2009. Ispytaniya aviatsionnykh dvigatelej (Aircraft engine testing). *Mashinostroenie, Moskow*.

JSC "Kuznetsov", 2019. *Accessed* January 10. http://www.kuznetsov-motors.ru/en.

NUMECA, 2008. User Manual AutoGrid5 Release 8.4, *NUMECA.inc., Belgium*.

NUMECA International The Nonlinear Harmonic module, 2019. *Accessed* January 10. https://www.numeca.com/product/nlhmethod.

Samara University, 2019. *Accessed* January 10. https://ssau.ru/english/.

Venediktov, V.D., Granovsky, A.V., 1990. *Atlas eksperimental'nykh kharakteristik ploskikh reshetok okhlazhdaemykh gazovykh turbin (The Atlas of Experimental Performances of Cooled Gas Turbine Blade Cascades),* CIAM.

Vilmin, S., Lorrain, É.,Tartinville, B., Capron, A. and Hirsch, Ch., 2013. The nonlinear harmonic method: from single stage to multi-row effects Int J of Computational Fluid Dynamics 27(2): 88-99.

Zrelov, V.A., 2002. Otechestvennye GTD. Osnovnye parametry i konstruktivnye skhemy. CHast' 2 (Domestic GTE. Key parameters and design schemes. Part 2). *SGAU, Samara*.