# NUMERICAL AND EXPERIMENTAL INVESTIGATIONS OF SINGLE-FLOW AND BYPASS-FLOW FANS

Victor MILESHIN, Igor OREKHOV, Sergey PANKOV

Central Institute of Aviation Motors (CIAM)
2 Aviamotornaya St., 111116, Moscow, Russia

## SUMMARY

The paper presents numerical and experimental investigations a bypass fan model and a ducted counter-rotating fan model. The method of computing bypass fans performances within RANS is developed. The results of the computation are compared with the experimental results. The objective of computations of counter-rotating fans is experimental aerodynamic and acoustic investigations before tests under the European VITAL project. Based on previous studies of a small-scale single-flow fan model, CIAM developed a large-scale by-pass flow fan model with a standard configuration and four booster stages and started its experimental investigations. The presented calculations of flow parameters and integral performances are carried out to make a choice of the best versions for designing as well as expert examination of high-efficient low-noise fan models proposed for tests at a special test facility.

## ABSTRACT

When designing the fans, all specific features of their operation in actual conditions should be taken into consideration. The effect of bypass ratio on local flow characteristics and integral fan performances is a time-consuming task. This work proposes a method for calculating the characteristics of bypass-flow fans on the basis of through-flow calculations of steady 3D viscous flow within RANS using a modified implicit version of S.K. Godunov's scheme [1]. A value of specified static pressure in outlet sections of inner and outer ducts is used as the boundary condition that is very similar to application of two independent throttles in each duct during tests.

Steady flows in blade channels of standard dual-flow fans and «VITAL» ducted fans as well their integral performances are calculated. Rotating and stationary blade rows are calculated in the «mixing plane» approximation [2-6]. The results of numerical and experimental investigations of two fan model versions - CRTF1 and CRTF2 under the European VITAL Program are presented [7]. A perfect coincidence between calculated and experimental data is shown. At present, the method gives a potential for calculations of ducted and unducted counter-rotating fans as well as several fan stages in the flow passage with flow splitting between ducts. OGV, struts and pylons can be taken into account in the outer duct; IGV, transition ducts, struts and several stages, e.g. a booster compressor, HPC or LPC can be taken into account in the inner duct.

The numerical and experimental investigations are used for C179-1 single-flow stage (its rotor manufactured as a «blisk» is a scaled down model-prototype of a dual-flow fan for an advanced bypass engine) and for C179-2 bypass stage – the scaled down version of the fan including 4 booster stages. In both cases the calculations show a good agreement between calculated data and test results. A number of turbulence models, including a two-parameter differential «k-ω» model, are used in calculations for closure the system of equations. Calculations of flow parameters and fan performances are carried out with the purpose of making a choice of optimal versions for designing as well as expert examination of model stages proposed for tests.

**Nomenclature**

| | | |
|---|---|---|
| CRTF | – | counter-rotating turbofan |
| D | – | blade tip diameter, m |
| G | – | mass flow, kg/s |
| IGV | – | inlet guide vane |
| Ma | – | Mach number |
| m | – | bypass ratio |
| $M_2/M_1$ | – | torque ratio |
| n | – | rotational speed, rpm |
| OGV | – | outlet guide vane |
| $P^*$ | – | Total pressure, Pa, kPa |
| R1 | – | first rotor |
| R2 | – | second rotor |
| RANS | – | Reynolds averaged Navier-Stokes |
| $T^*$ | – | Total temperature, K° |
| $U_C$ | – | tip speed, m/s |
| Z | – | number of blades in a row |
| $\Delta K_{st}$ | – | stall margin |
| $\eta^*_{ad.}$ | – | adiabatic efficiency |
| $\pi^*$ | – | total pressure ratio |
| σ | – | total pressure recovery |

**Subscripts**

| | | |
|---|---|---|
| ad. | – | adiabatic |
| air | – | air |
| cor | – | corrected |
| f | – | fan |
| st | – | stage |
| I | – | core duct value |
| II | – | bypass duct value |

# INTRODUCTION

Development and implementation of high-efficient low-noise fans with wide-chord blades aiming at a decrease in noise emission, an increase in efficiency and an improvement of stall margins is a present-day tendency in aviation. Noise level of fans and their efficiency depend on tip speeds of rotating blades. Within the last 15 years the tip speed of fan blades has been reduced from $U_C = 450$ m/s to $U_C = 400$ m/s with the purpose of a noticeable improvement of acoustic characteristics and efficiency. The fan delivery rate has been increased with a slight decrease in pressure ratio and an increase in efficiency up to a record value - $\eta^*_{ad} = 0.92 \div 0.93$. A reserve for a further decrease in tip speed to meet exacting requirements of ICAO Chapter IV can be provided by counter-rotating fans (CRF). For example, $\pi^*_f = 1.5$ (total pressure ratio) in the CRF can be achieved by a noticeable decrease in the tip speed down to $U_C \sim 280$ m/s ($\Delta U_C \sim 120$ m/s) as compared with a standard single-row fan. The potential for eliminating or minimizing the noise level can be realized but it is necessary to study and analyze in details the gasdynamics of unsteady rotor interaction. An additional decrease in specific fuel consumption with an increase in bypass ratio and a decrease in total pressure ratio and rotational speeds of the blades are possible for open rotors providing higher flight efficiency at lower flight speeds. The task of European VITAL program (EnVIronmenTALly Friendly Engine) is the development of an advanced "clean and quite" engine. The project is implemented by leading European engine manufacturers. The project coordinator is Snecma (France) [7]. Many European companies, including Snecma, DLR (Germany), CIAM (Russia) take part in the development of a ducted CRF consisting of 2 blade rows with counter-rotation provided by direct driving from two independent shafts.

# CALCULATIONS OF STEADY FLOWS AND CRF INTEGRAL CHARACTERISTICS AND COMPARISON WITH TESTS IN C3-A ANECHOIC TEST CHAMBER (CIAM)

The CRF demonstrator model with a booster stage (Fig.1) was developed and manufactured for experimental investigations in the anechoic chamber of C-3A acoustic test facility.

The fan model with an antisurge device at the fan inlet was installed on a counter-rotating shafting. The test facility is designed for measurements of acoustic characteristics in the anechoic chamber both in front and rear hemispheres at the distance more than ten times of the fan diameter. The fan experimental model diameter was 0.5588 m (22”). The configuration of the model casing depends on a test program. A vaned throttle and cases with rigid smooth walls are installed at the flow passage outlet for measurements of the fan aerodynamic characteristics. Replaceable nozzles different throat cross-sections ($D_n$=595, 603, 618 mm) at the flow passage outlet and cases with acoustic treatment are used for acoustic tests. Bypass ratio of the fan is controlled by an inner duct throttle.

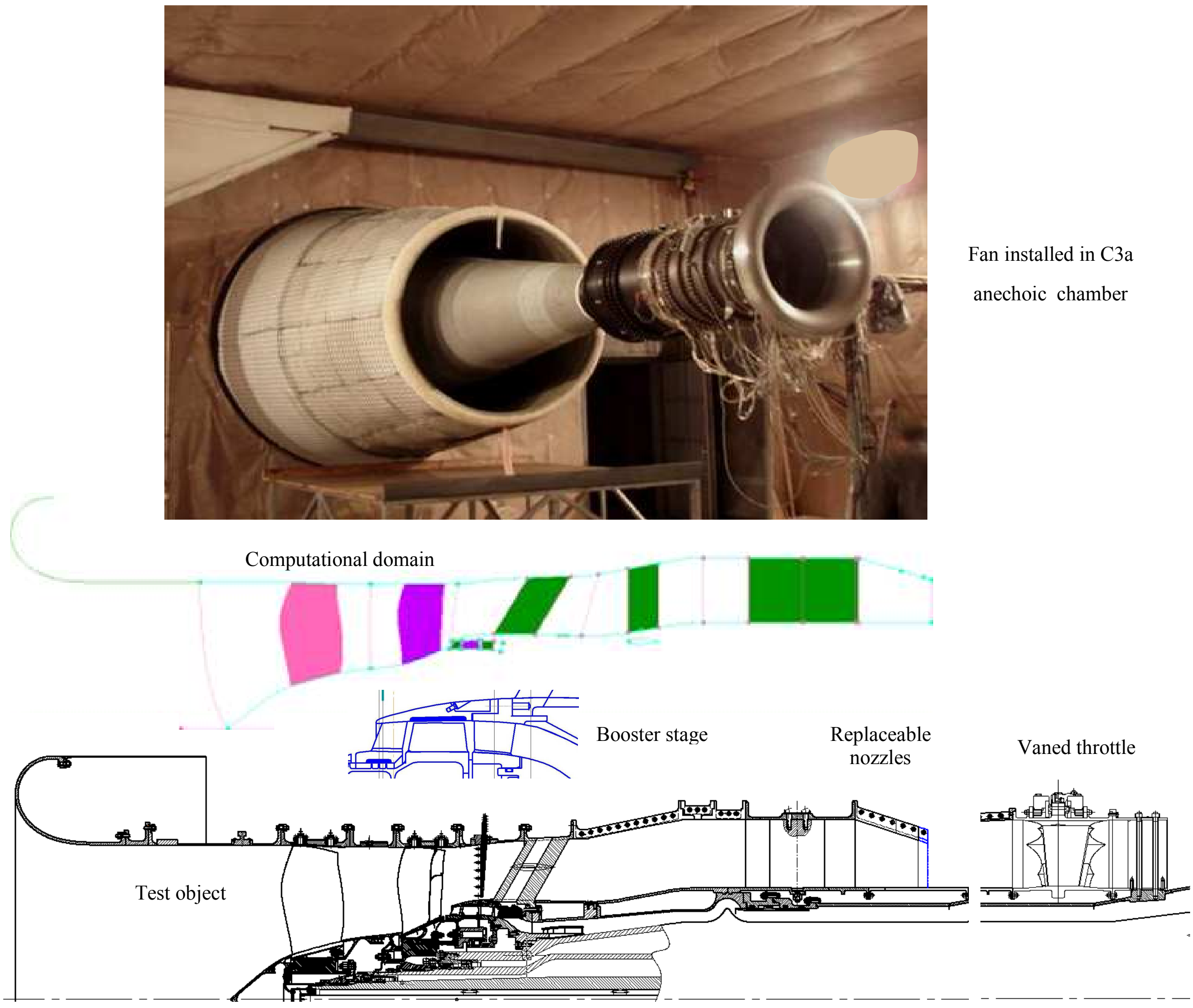



*Fig.1. Longitudinal section of CRF model*

Aerodynamic and acoustic tests of the first CRF version (Snecma) were completed. Two additional versions (CIAM, DLR) were tested because it was necessary to optimize $n_2/n_1$ and work distribution (or $M_2/M_1$ torque ratio) between R1 and R2 rotors, find an optimal distance between them and optimal number of blades in rows (Z=10,14 and Z=9,11). Thus, three fan versions were designed and tested. Blades of the second and third (DLR) fan versions as well as the original fan version (Snecma) were made of titanium alloy. The second fan version (CIAM) takes into account specifics of full-scale fan blades made of composite materials: maximum profile thickness and edge thickness are increased. Additionally, number of blades in the third version (DLR) was reduced (Z=9, 11). The second and third fan versions were optimized in such a way to provide the same or better aerodynamic parameters as compared with the original fan. Additionally, a standard single-stage fan (Fig. 2) was tested for comparison [7, 8].

3-D viscous flow computations were completed in «mixing plane» approximation. Unsteady interaction of vanes is not taken into account. Flow is averaged by a pitch in the row-to-row axial clearance to find steady flow solutions. In this case the entropy increases that simulates losses when smoothing pitch non-uniformity because of mixing. This procedure is "non-reflecting" as a shock waves falling to the boundary does not reflect back to the computation domain. Distributions of total pressure and temperature (ISA 101325 Pa, 288.15°K), an angle between the velocity vector and the meridian plane as well as an angle between the velocity vector projection onto the meridian plane and the fan axis are specified as boundary conditions at the inlet; periodicity conditions on periodic boundaries and wall functions on impermeable surfaces shall be observed.

Calculations of steady flows and integral performances were completed before tests in support of aerodynamic and acoustic investigations. Flow splitting between ducts, a booster stage and all structural components of the outer duct, struts and pylons right up to the nozzle exit section were taken into consideration (Fig.1, Fig.2).

The algebraic or differential two-parametrical turbulence models used for calculations of turbulent viscosity $\mu_t$. Dimensionless wall distance $y^+=10\div15$. The static pressure distribution at the hub is specified at the outlet, and the pressure distribution along the flow passage height is found from the condition of approximate radial equilibrium with account of residual flow swirling (can be 0). The computations are based on simple computational grids as parallel to the front and meeting together in nodes of periodicity lines. The computational grid has a concentration near to impermeable surfaces for the boundary layer and near to blade leading and trailing edges for correct profiles description.

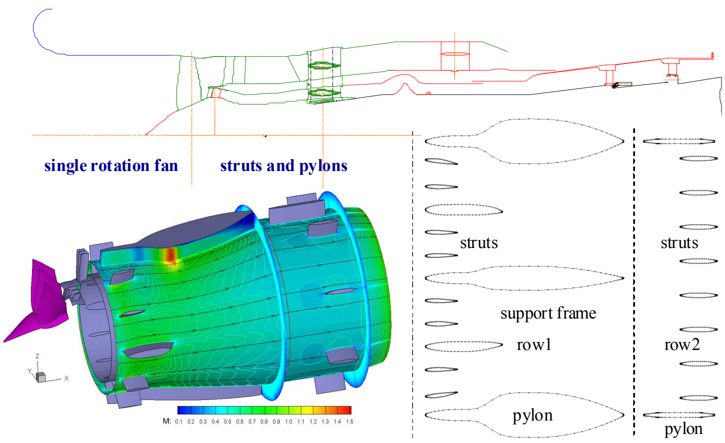



*Fig. 2. Conventional fan model, struts and pylons.*

Blade strains under action of gas and centrifugal forces were approximately taken into account. Blade shape and tip clearances were found by calculations of strain-stress state of blades at $\overline{n}_{cor}$=100% design point in cruise. "Hot" R1 tip clearance is approximately 0.35mm and R2 is 0.27mm. The computational grid exceeds 1 million cells. There are about 100 cells along profiles on suction and pressure sides. There were ~10 computational grid cells in the tip clearance. Fan performances were calculated for all values of rotation speeds - $\overline{n}_{cor}=50\div110\%$ within a wide range of operating conditions - from flow choking to the surge line (Fig. 3). Thermocouple anemometers were aligned with the computed flow angle between rotors to obtain correct measurements of the velocity pulsations. Measured distributions of parameters along the channel height were compared with the calculation (see Fig. 4).

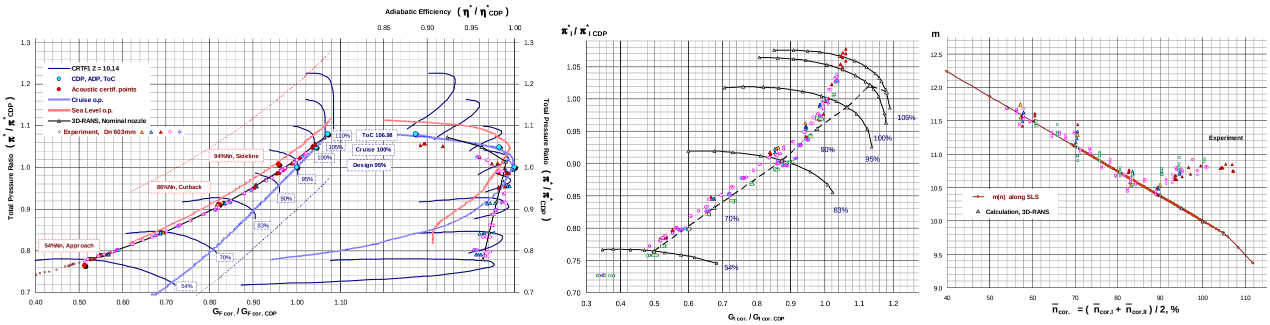
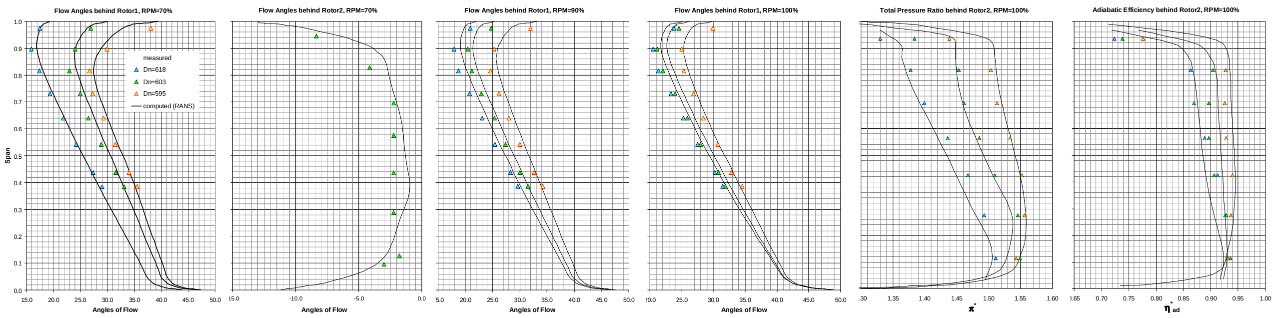


*Fig. 3. Comparison of CRF calculated parameters and test data.*



Flow angles at R1 and R2 outlets, $\bar{n}_{cor}$=70% | Flow angles at R1 outlet, $\bar{n}_{cor}$=90, 100% | Total pressure ratio and adiabatic efficiency at R2 outlet, $\bar{n}_{cor}$=100%

*Fig. 4. Comparison of calculated and measured distributions of parameters at CRF R1 and R2 outlets.*

The area of outlet sections for 3 nozzles was calculated for acoustic tests. The first nozzle with maximum exit section (open) was designed for measurements in operating modes on right branches of the characteristics. The second base or nominal nozzle was required for investigations in conditions slightly lower the ground operating line at the points of acoustics certification. The third nozzle was required for tests in conditions on the left branches of the characteristics near to the surge line. Exit jets, instrumentation, joints, etc. were not taken into account in the calculation. Therefore, it was recommended to start testing with an open nozzle. Fan operation at low rotational speeds with the open nozzle was in the area of high efficiency that was very useful for the first cold start. Based on these results, other two nozzles were cut down.

When testing, the dependence between bypass ratio and rotational speed was supported as known for the ground operating line. In calculation of nozzles-fan matching lines the backpressure value at the outlet of the inner duct booster stage was chosen in such a way to support $m = m(\bar{n}_{cor})$ relationship. The inner duct performances were calculated for the nominal nozzle installed in the outer duct.

The fan with not variable blades can be controlled by changing $n_2/n_1(\bar{n}_{cor})$ ratio. The basic $n_2/n_1(\bar{n}_{cor})$ law was observed and 2 additional laws for an increase and a decrease in rotor 1 rotational speeds - «$n_1$+5%» and и «$n_1$-5%» were studied. Changes in $n_2/n_1$ ratio resulted in redistribution of work between rotors and a different relationship between torque ratio and rotational speed - $M_2/M_1(\bar{n}_{cor})$. By choosing the $n_1$ value it is possible to provide a required law, e.g. to support $M_2/M_1(\bar{n}_{cor})$=const for fans driven by a differential gear. When supporting the basic law [7] - $n_2/n_1(\bar{n}_{cor})$, $M_2/M_1$ ratio varies along the characteristics. In tests with replaceable nozzles in the outer duct each nozzle provides its own $M_2/M_1(\bar{n}_{cor})$ dependence. The fan control by changing $n_2/n_1(\bar{n}_{cor})$ was demonstrated in tests with a nominal nozzle installed at the outer duct outlet.

The comparison showed that calculated results are in good agreement with experimental data, but calculated values of $\pi^*_f$, $G_{f\,cor}$ at $\bar{n}_{f\,cor}>0.9$ were slightly higher than measured values. Therefore, the mathematical model can be modified and the fan can be improved by operational development of the experimental model.

Accumulated experience in designing and test results can be used in developments of new engines for short- and medium-range aircraft.

## 3D FLOW CALCULATION FOR A STAGE OF AN ADVANCED FAN

The C179-2 stage is a model of a dual-flow gearless fan and 3-4 booster stages of an advanced turbofan for civil aircraft. This work was aimed at creation a scientific and technological potential for optimal designing and mathematical modelling of processes as well as experimental investigations of aerodynamic and acoustic characteristics of new fans at special test facilities. Initially designed rotor blades of a full-scale fan were straight (without a sweep) and calculations showed underestimated parameters of inner (LPC, a rotor hub and booster stages) and outer ducts. In the final fan model with $D_f$=700 mm the rotor blades were designed with variable sweep along the height; number of rotor blades and stator vanes were decreased in all stages in the booster compressor of the model as compared with the full-scale fan. It is worth noting that a small-scale single-flow fan model ($D_f$=400 mm) was designed, manufactured and tested. Calculated and measured characteristics of the small-scale model were in good agreement and the stage provided high values of maximum efficiency within the total operation range ( $\bar{n}_{cor}>0.5$ ). Acoustics and sound-absorbing properties of various coatings were also studied for the stage [5].

Tip diameter of the model is 700 mm, corrected tip speed of rotor blades at $\bar{n}_{cor}=1.00$ is $U_{C\,cor}$=404.5 m/s. Design values of the fan model are the following: total pressure ratio in the outer duct – $\pi^*_{II}$=1.56; total pressure ratio in the inner duct – $\pi^*_{II}$=2.76; bypass ratio of the fan – m=8.4.

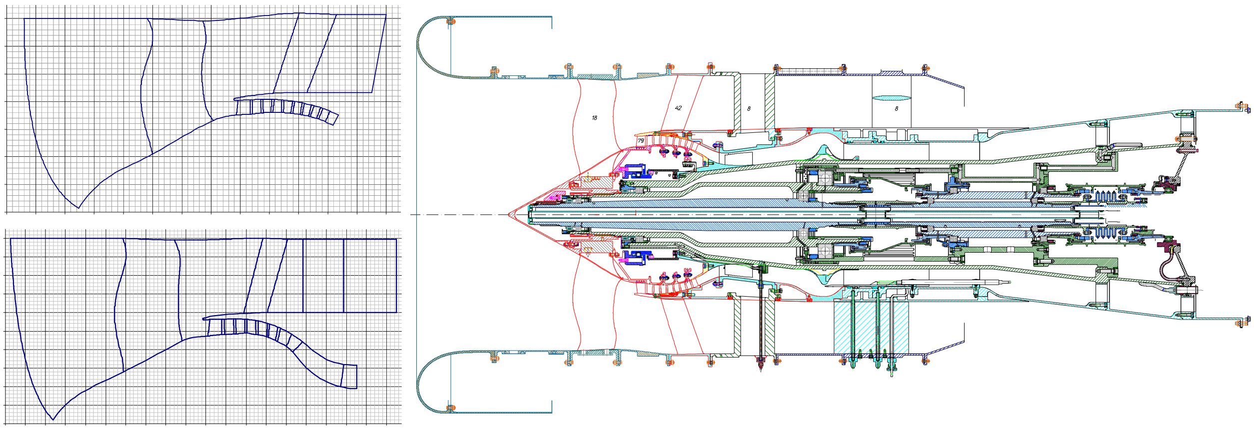
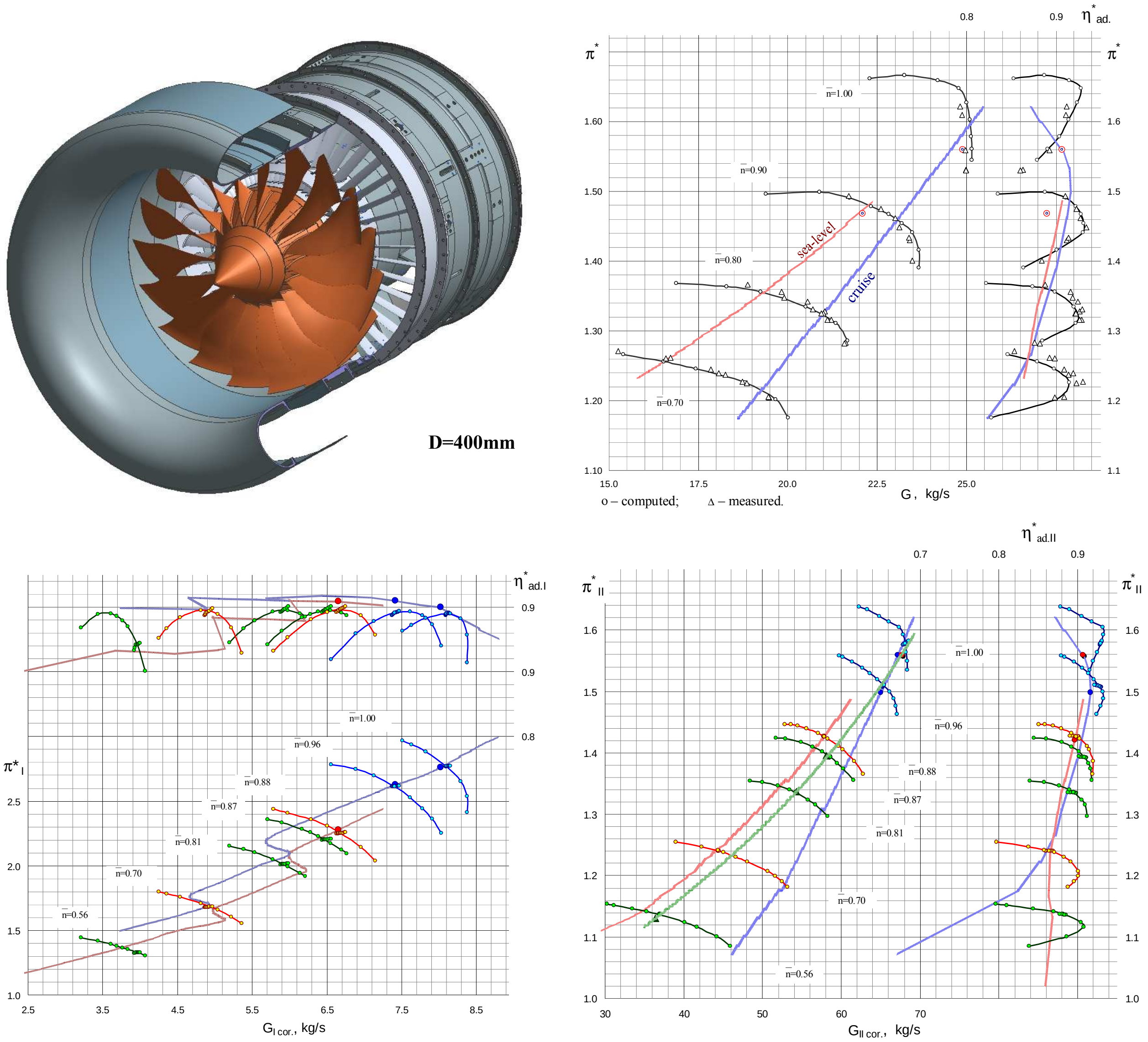


*Fig. 5. Longitudinal section of C179-2 fan model. Computational domain. Fan, 3-4 booster stages, struts, outer duct.*

In addition to this operating mode, flows in cruise with maximum efficiency value in the outer duct ($\eta^*_{ad\,II}$ = 0.917) were calculated. Flow calculations were carried out for 2 take-off modes: $\bar{n}_{cor}$=0.882, $U_{C\,cor}$=357 m/s and $\bar{n}_{cor}$=0.704, $U_{C\,cor}$=285 m/s. Moreover, calculations of 3 modes that are important in view of acoustics certification were carried out. The following operating modes were chosen for calculations of acoustic characteristics: $\bar{n}_{cor}$=0.867, $U_{C\,cor}$=351 m/s; $\bar{n}_{cor}$=0.814, $U_{C\,cor}$=329 m/s; $\bar{n}_{cor}$=0.556, $U_{C\,cor}$=225 m/s. Thus, three-dimensional steady viscid flows of compressible gas as well as total characteristics of C179-2 experimental bypass fan model with booster stages were calculated for 7 modes: $\bar{n}_{cor}$=0.56, 0.70, 0.81, 0.87, 0.88, 0.96, 1.00. Additionally, calculations in other operating conditions of the stage were also completed.




**D=700mm**

*Fig. 6. C179 fan model. Calculated and test data of the fan model with $D_f$=400 mm.*
*Calculations of outer duct performances for the fan model with $D_f$=700 mm.*

The shape of blades in calculations corresponds to their stress-strain state under action of gas and centrifugal forces in the design mode. The calculations were carried out with approximate account of a cone and a tip clearance between rotating rotor blades and a stationary outer casing; the tip clearance was approx. 0.4 mm; the clearance between rotor blades of the booster compressor and the inner duct of the splitter was 0.3 mm. Tip clearances between rotating rotor blades and a stationary casing as well between S1, S2, S3 one-end fixed vanes and a rotating drum-type hub were equal to 0.3 mm. IGV and S4 vanes were two-end fixed (no tip clearances, no hub rotation).

The computational domain covers one blade channel in each of 11 rows. At first, values of static pressure (i.e. boundary conditions at outer and inner duct outlets ) in calculations were chosen for each operating mode in such a way to support $G_{f\,II\,cor}$, $\pi^*_{f\,II}$ and $G_{I\,cor}$, $\pi^*_{I}$ very close to design values on the corresponding operating lines.

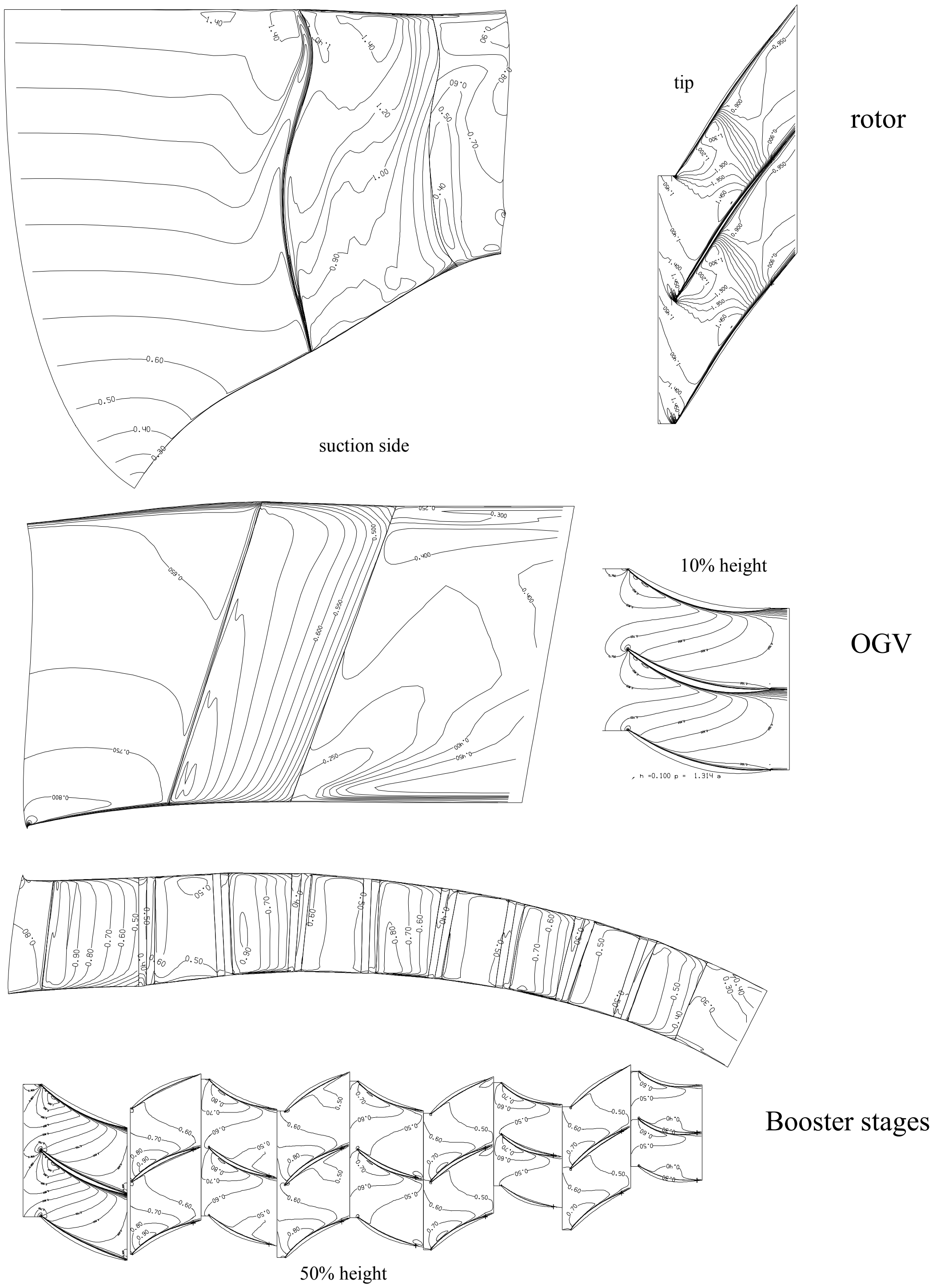



*Fig. 7. Stage C179-2 - fan model for a bypass engine.*
*Close to the design mode $\bar{n}_{cor}$=100%. 3D-RANS,*
*Mach-number isolines*

Then, performances of inner and outer ducts were calculated for 7 operating modes: $\bar{n}_{cor}$=0.56, 0.70, 0.81, 0.87, 0.88, 0.96, 1.00. Only static pressure in 1 duct was variable in calculations of performances; static pressure in another duct was kept constant. Bypass ratio of the fan - $\boldsymbol{m}=G_{f\,II}/G_{f\,I}$ was variable depending on changes in $G_{f\,I\,cor}$ or $G_{f\,II\,cor}$ along the characteristics.

Fig. 6 shows calculated performances of a single-flow stage with $D_C$=400 mm and a dual-flow stage with $D_f$=700 mm as found by independent throttling of outer and inner ducts of the stage. Red lines in Fig. 6 correspond to ground operating modes, .i.e. show the ground operating line $M_{fl}$=0, $H_{fl}$=0 and two characteristics, when the static pressure values are chosen in such a way to support the bypass ratio equal to specified values near to the ground operating line. Blue lines correspond to high-altitude conditions - $Ma_{fl}$=0.8, $H_{fl}$=11 km and show 2 characteristics when static pressure values are chosen in such a way to keep the bypass ratio equal to specified values very close to the altitude operating line. Green color shows 3 characteristics passing through the points corresponding to acoustics certification modes as well as the operating line that can be found as a line of combined operation of the fan and the nozzle with a required cross-section that is installed at the outer duct outlet. As shown, maximum efficiency of the outer duct in calculations was slightly overestimated and efficiency of the inner duct (for total LPC) was lower than the required values.

Flow with a shock located in blade channels near to trailing edges is observed in tip sections of the fan rotor. Flow velocity in tip sections on suction sides exceeds M=1.45. In bottom sections because of positive incidence angles flow at first accelerates in suction waves at the inlet and then decelerates in shocks at the outlet. Fig. 7 shows Mach number distribution in the design operation condition of the fan: $\bar{n}_{cor}$=11036 rpm, $U_C$=405 m/s ($Ma_{fl}$=0.8, $H_{fl}$=11 km).

Flow velocity in the outer fan duct on the OGV can't reach sonic speed even at the bottom of blade channels on suction sides of guide vanes. Flow velocities in the inner fan duct on the HPC IGV suction sides are slightly higher than M=1; flow in blade channels of rotors and stators of booster stages is subsonic.

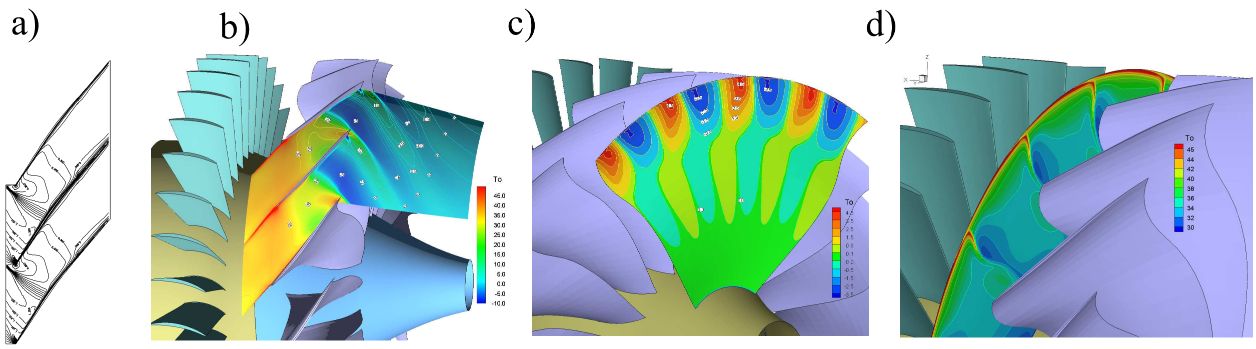



*Fig. 8. Stage C179-fan model for a bypass engine. Close to the operating mode $\bar{n}cor$=90%. 3D-RANS, T* distribution*

*a) Mach number isolines in the rotor blade channel near to the tip*
*b) T* distribution in rotor tip sections*
*c) T* at the Stage inlet*
*d) T* in the center of the axial clearance between rows of rotor blades and stator vanes.*

Flows at lower rotational speeds (≤ $\bar{n}_{cor}$=0.90) in rotor tip sections for both single-flow and bypass versions are observed with alternating detached shocks and suction waves at the inlet. Maximal value of Mach number in tip sections on the suction sides before chock is equal to M=1.35 (Fig. 8). The periodic system of detached shocks and suction waves results in a high static pressure difference at the inlet that is the primary source of noise emitted in the front hemisphere in these operating conditions. In this case $T^*$ total temperature ($h^*$ enthalpy in absolute motion) increases in shocks (rotating with the rotor) and decreases in suction waves: $\frac{Dh^*}{Dt} = \frac{1}{\rho}\frac{\partial p}{\partial t}$ [9]. Therefore, periodically alternating regions of increased and decreased «$T^*$» total temperatures (see Fig. 8) are rotating with the rotor at the stage inlet. It is clear that «$T^*$» pulsation frequency is equal to blade passing frequency and the pulsation intensity decreases with a distance from blades and changes along the height. Fig. 9 shows distributions of «$T^*$» in tip sections of the rotor blade channels for operation of the stage with detached shocks at $\bar{n}_{cor}$=90%. Detailed distributions of total temperature in inlet sections give the evidence that the difference between T* and the mean value at the tip is approx. ±5° at the distance from the leading edges equal to the blade chord. Total temperature distribution in the section located at the rotor outlet gives the evidence that there is an increase in maximum total temperature near to the outer casing and in blade wakes and the difference between T* and the mean value is greater than Δt~ ±5°. There is an increase in the mean value of total temperature in the stage rotor from the inlet to the outlet by 35° ($\pi^*_R$= 1.47) in this operating mode. Today, these values of total temperature pulsations at the stage inlet as well as spectrums and values of static pressure pulsations can be measured [10].

Steady flow fields in relative motion that are used as input data in 3D calculations of unsteady disturbance fields with the purpose of numerical simulation of noise generation with account of rotor-stator interaction were more accurately calculated to find unsteady and acoustic fan characteristics by using a very detailed computational grid at the following points of the characteristics: $\overline{n}_{cor}$=0.867, $U_{C\ cor}$=351 m/s; $\overline{n}_{cor}$=0.814, $U_{C\ cor}$=329 m/s; $\overline{n}_{cor}$=0.556, $U_{C\ cor}$=225 m/s corresponding to acoustics certification modes. The detailed computational grid was almost uniform in axial and radial directions without condensation at edges but with condensation across the channel towards blade surfaces to take into account the boundary layer but without account of the tip clearance. Distributions of parameters of steady flows in relative motion calculated in this computational grid were used as input data in calculations of propagation of 3D unsteady disturbances in steady non-uniform 3D viscous flow by the method described in work [11]. But they were preliminary interpolated in the computational grid that was uniform in the crosswise direction.

## CONCLUSIONS

The calculation procedure for bypass fans on the basis of 3D through-flow calculation of viscous flow within RANS by using S.K. Godunov's modified implicit scheme is developed. The value of specified static pressure in outlet sections of outer and inner ducts is used as the boundary condition. The solution concept is very similar to the method used in experimental investigations of bypass fans when two independent throttles are used in each duct.

Flow parameters and performances of CRF and bypass standard-type fans are calculated with account of booster stages, struts and pylons in the outer duct right up to the nozzle exit section.

The comparison of calculated and experimental integral characteristics of single-flow fans within a wide operation range from $\overline{n}_{cor}$ = 0.7 to $\overline{n}_{cor}$ = 1.0 shows good agreement of results. The difference in values of key parameters is not above 1 % with the exception of $\overline{n}_{cor}$=1.0, where the difference in efficiency is 1.5 %; specified values of efficiency and air flow in this mode are reached but at a higher total pressure ratio (by ~3 %). Measured maximum values of efficiency are equal to $\eta^*_{ad.\ max}$=0.92…0.93 at intermediate values of rotational speeds ($\overline{n}_{cor}$ = 0.5 … 0.95).

Comparison of calculated and experimental integral characteristics of CRFs shows good agreement of results. But the difference between calculated and measured values of air flow and total pressure ratio at $\overline{n}_{cor}$>0.9 exceeds 2 %. Experimental values of these parameters are slightly lower than calculated values. Therefore, the mathematical model can be modified and the fan can be improved by operational development of the experimental model. Measured distributions of flow angles, total pressure ratio and efficiency along the channel height are very close to distributions found by calculations.

# BIBLIOGRAPHY

[1] S.K. Godunov, A.V. Zabrodin, M.Ya. Ivanov, A.N. Kraiko, G.P. Prokopov –*Numerical solution of multidimensional gasdynamic problems*. M., Nauka (Science), **1976**, 400 pp.

[2] Lisa Brilliant, Stanley Balamucki, George Burger, Yuan Dong, and Charlie Lejambre – *Application of multistage CFD analysis to low pressure compressor design*. GT2004-54263, ASME-2004, Vienna, Austria, June 14-17, **2004**.

[3] V.I. Mileshin, I.K. Orekhov, S.V. Pankov, V.A.Panin – *Computation and investigation of flow in counter-rotating propfans including reverse thrust regimes*. XVI International Symposium in Air-Breathing Engines, Cleveland, Ohio, USA, August 31 –September 5, ISABE-**2003**-1169.

[4] I.A. Brailko, V.I. Mileshin, M.A. Nyukhtikov, S.V. Pankov, A.A. Rossikhin – *3D computational analysis of unsteady and acoustic characteristics of high by-pass ratio counter-rotating fan models*. ISABE-2005-1186, September 4-9, Munich, Germany, **2005**.

[5] V.I. Mileshin, I.K. Orekhov, S.V. Pankov – *Numerical and experimental investigations of bypass fan characteristics.* ISABE-2007-1138, September, 9-1, Beijing, China, **2007**.

[6] V.I. Mileshin, M.A. Nyukhtikov, I.K. Orekhov, S.V. Pankov, S.K. Shchipin – *Optimization of counter– rotating fan blades based on 3D inverse problem Navier-Stokes solution method with the aim of tonal noise reduction.* Proceedings of GT2008 ASME Turbo Expo 51173, Berlin, Germany, June 9-13, **2008**.

[7] Jérôme Talbotec, Michel Vernet – *Snecma counter-rotating fan aerodynamic design logic & tests results.* ICAS, Nice, France, 19 - 24 September, **2010**.

[8] Timea Lengyel, Dr. Eberhard Nicke, Dr. Klaus-Peter Rüd, Dr. Reinhold Schaber – *Optimization and examination of a counter rotating fan stage – the possible improvement of efficiency compared with a s*ingle *rotating fan*. ISABE-2011-1232 Gothenburg, Sweden, September 12-16, **2011**.

[9] E.M. Greitzer, H.P. Hodson, T.P. Hynes, C.S.Tan – *Physical interpretation of stagnation pressure and enthalpy changes in unsteady flow*. Proceedings of ASME Turbo Expo 2009: Power for Land and Air GT2009-59374, Orlando, Florida, USA, June 8-12, **2009**.

[10] N. N. Ledovskaya, S.V. Pankov, E.P. Gladkov, A.N. Mercurev, A.M. Gorbatchev. – *Numeric and Experimental Investigations of Unsteady Flow Structure in Compressors with Application of Up-Date Techniques.* Tekhnika Vozdushnogo Flota (TVF, Aviation Science and Technology) (ISSN: 0868-8060), vol. LXXXIV, №1(698), **2010**, p. 48-63.

[11] A.A. Rossikhin, S.V. Pankov, I.A. Brailko, V.I. Mileshin – *Numerical Method for 3D Computation of turbo machinery tone noise.* Fan 2012-035, Senlis, France - April 18-20, **2012**.

E-mail: mileshin @ciam.ru, sergey-pankov@mail.ru

Keywords: counter-rotating fan, aerodynamics, fan design, fan aerodynamic tests