APPLICATION OF 3D CFD METHODS TO THE ANALYSIS OF LARGE PARTICLES INGESTION TO THE ENGINE INLET LOCATED NEAR THE AERODROME SURFACE

S.Yu. Krasheninnikov, D.E. Pudovikov
Central Institute of Aviation Motors, Moscow, Russia

The low under-wing engine mounting typical on the modern civil airplanes causes the initiation of the powerful vortex-type flow at the inlet of engine when it is in operation. This vortex originates near the ground surface and picks up from it particles, which are ingested into the air intake.

The sizes of particles, which can be carried by vortex, depend on the vortex intensity. The particles transportation is a very complicated process that is a result of interaction of different complex phenomena. Among them are: particles shifting and lifting, their motion in severe aerodynamic conditions, and others.

This report suggests the results of numerical analysis of a vortex-type flow, which begins during operation of a modern civil airplane turbofan engine mounted close to the airfield surface. The influences of different conditions on the flow pattern near ground surface and at engine inlet are analyzed. Among them are: cross wind, engine operating modes, airplane motion, pod height above ground and presence or absence of exhaust jet.

The investigations performed permitted to determine the factors, which influence the vortex initiation and intensity. The comparison of flow patterns during the engine operation with and without an exhaust jet have shown that the jet ejecting effect had a considerable influence on ingestion of the ambient air into the engine inlet. The analysis of particles motion under conditions considered allowed to determine the main parameters, which influence the vortex flow ability to carry big particles to the engine inlet.

Problem formulation and calculation method

To analyze the process of particle ingestion into the air intake duct, it is necessary to have a full enough notion of air motion around the intake near ground surface.

Complexity of measurements in a vortex flow is a reason of insufficiency of existing experimental data. The available results of practically isolated measurements of air flow velocity near the surface [1] cannot reveal the effect of flow acceleration in vortex that could explain picking up and lifting big particles from the ground surface. Therefore, it is necessary to supplement the experimental data with the results of adequate enough calculations.

At present, such calculations can be made on the basis of numerical integration of Reynolds equations with a turbulence model. A flow field with vortex and without it, the full 3D distribution of velocity components, the values of which can be considered corresponding to the real values of air velocity, levels of pressure variations in a vortex are possible to obtain from such calculations. The flow parameters will be approximately the same as for a real flow.

The time relaxation method for solving Navier-Stocks system of equations averaged by Reynolds was used for description of 3D flows of compressible viscous gas.

Various types of boundary conditions were used in calculations. The following condition was assigned for the surfaces flown over walls:

$$\boldsymbol{q}_w=0, \quad (\nu_t)_w=0,$$

where $\boldsymbol{q}_w$, $(\nu_t)_w$ – velocity vector and turbulent viscosity on the wall. In addition to the above conditions, setting the zero values for normal derivatives of the flow parameters on the boundaries, through which the gas enters the calculation region is necessary. Values of ν_t were assigned to inlet boundaries, too.

The uniform flow parameters over the entire calculation region should be specified as initial ones for all the flow patterns. ν_t-90 model was used for description of turbulence [2].

The solution of the equations set with the assigned boundary and initial conditions is found during the time relaxation process. The finite-difference method after Godunov was used to determine the flow parameters at every time step [3].

The mathematical model of the test object included an air intake, a pod and a jet. The pod and intake outlines were associated with the modern engine with mixing of bypass and core flows and a bypass ratio of about 5. The “flow path” has the center body and surfaces perpendicular to the engine axis so that the boundary conditions set on these surfaces could satisfy in calculations the specified flow at the engine inlet as well as the total parameters and flow at the engine outlet.

The boundaries of the calculation region include: $z = const \equiv 0$ plane - airfield surface; $y = const$ plane – aircraft symmetry plane; $x = const$ planes, which are located at 30 inlet diameters before and 60 after the pod; the cylindrical surface over the pod placed at a distance not less than 40 inlet diameters from the pod.

The calculation of flow for the problem considered was carried out with simulation of different engine operating modes and values of cross wind velocity and aircraft movement speed. The influence of exhaust jet and pod height above ground on the flow formed near the ground surface and at the air intake inlet was investigated too.

The accuracy and adequacy of calculations were checked by special procedures. The calculation of flow in a jet was one of fidelity criteria for the calculations made. The flow parameters obtained in calculations were in good agreement with the well-known data for turbulent jets. In correspondence with [4], the jet axis bent towards the surface owing to the effect of jet attachment.

Results of flow pattern calculation

The typical results of flow pattern calculations are presented in fig.1. The lines correspond to flow lines, the gradient color field – to velocity distribution.

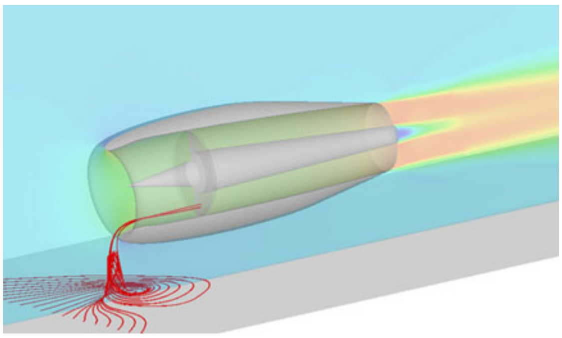

Fig. 1

The flow regimes with and without vortex were realized in calculations. It should be noted that for the case without vortex (fig. 2),

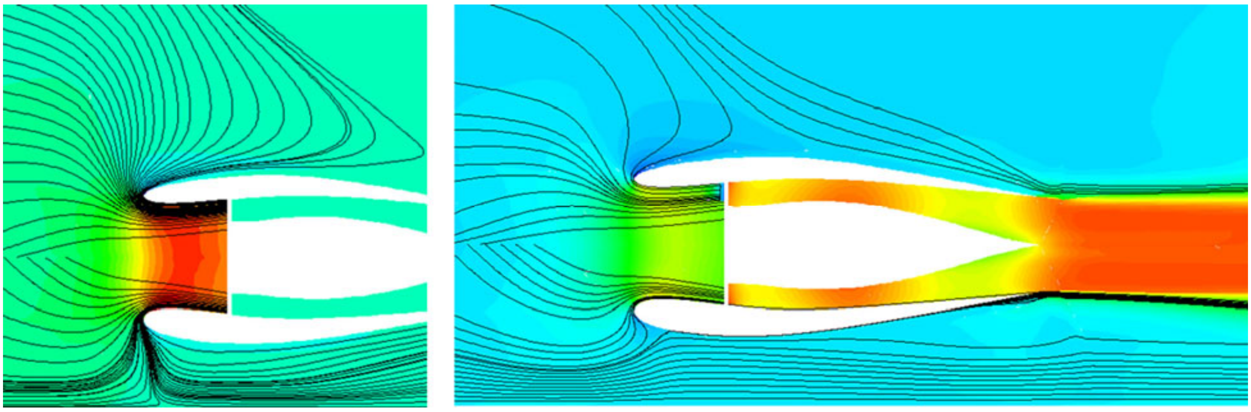

Fig. 2

the jet ejecting the ambient air has an explicit influence on the flow pattern. In this case, most of near-ground flow lines under the intake duct go to the jet. For the case with vortex, the influence of jet is not so explicit.

For the subsequent analysis of the velocity components distributions it is convenient to denote as U the value of velocity induced in the near-surface region. In a free space, the velocity values at distance H from the inlet may be assessed by equation

$$U/u_i = (1/2 \div 1/4)\ \ R_i^2 / H^2 ,$$

where factor 1/2 corresponds to small values of H/R_i, factor 1/4 – to $H/R_i \to \infty$. Here and hereinafter u_i is a velocity at the intake duct inlet, R_i is a duct radius, H denotes distance up to the center of duct inlet. If the surface is located at this distance, then the value of U can be used for estimation of the velocity induced near the surface.

Different variants of engine operating modes and external conditions, which will be used in the discussion of calculation results, are presented in the table.

Table

regime indication	▲	✕	●	■
with/without (+/-) exhaust jet	-	-	+	+
H/R_i	2.25	3.5	1.9	2.25

The question concerning influence of vortex initiation conditions on the vortex intensity were investigated during flow modeling by calculation. As was found, the asymmetry should be present in the original statement of the problem in order to obtain the vortex-type flow in calculations.

The influence of cross wind on vortex intensity was investigated. The calculations showed that if the velocity value of cross or longitudinal air motion is smaller than the value of induced velocity U, then the vortex intensity is practically the same (independent of it).

The results of these calculations were used in determining the distribution of different velocity components in vortex at various heights above the ground surface. Besides, the calculation results obtained for the modes with and without jet were compared.

Fig. 3 shows, as an example, the results of calculated velocity components distribution for various engine-pod height above the ground – both with and without the exhaust jet. The velocity components values are referred to the inlet velocity value, which corresponded to Mach number $\mathsf{M} \approx 0.5$. The values of the rotational component V_ϕ of the velocity vector, radial component V_r directed to the vortex center, and normal to the ground surface vertical component V_z are presented. Here R is a distance from vortex center line, z – height above ground. Designation $z \sim 0$ correspond to height value 10-15mm.

The rules of changing the maximum values of velocity components V_r and V_ϕ with re-

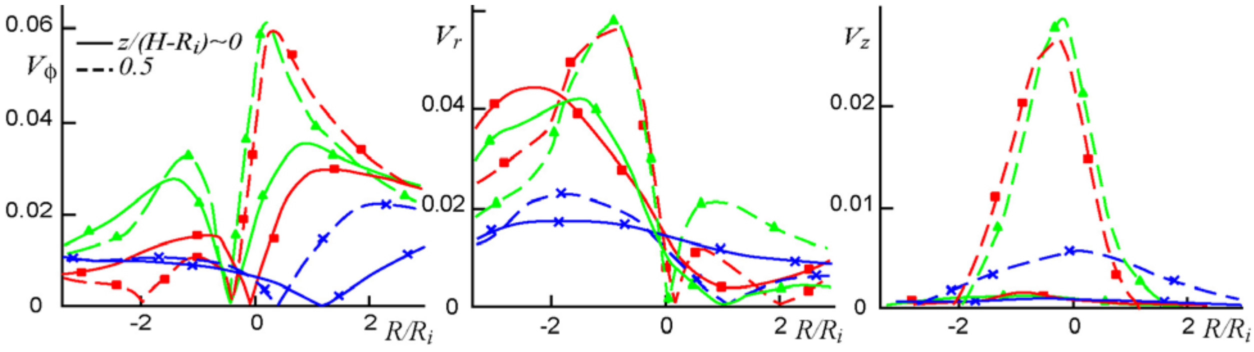

Fig. 3

spect to height above ground are given in fig. 4.

The calculation results demonstrate that velocity components V_r and V_ϕ near the ground surface and up to z less than $0.5R_i$ vary slightly, and their values are approximately the same as those of U. On the other hand, the vertical velocity component V_z increase monotonously with increase of height above z value.

The maximal value of rotational velocity component increases with height above ground, but not so fast as the vertical component.

As can be seen from the figures, the value of radial component V_r, depends on the presence of jet. This component characterizes the air motion to the vortex center. The intensity of this motion decreases from the jet side. The value of rotational component decreases too, especially in the near-surface layer. The upflow is practically independent of the jet presence. To characterize the intensity of vortex-type flow at various heights above ground surfaces

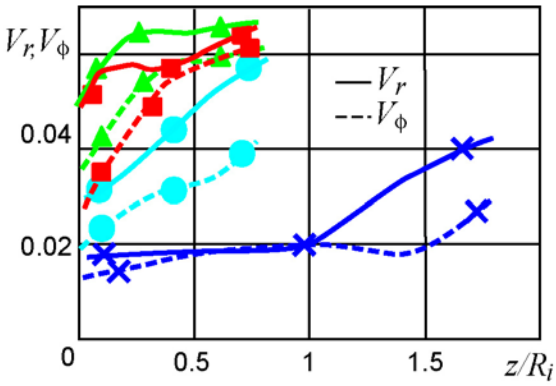

Fig. 4

as well as the dependence of this intensity on main parameters of the problem, let's take the value of non-dimensional circulation as Γ.

$$\Gamma = \frac{\oint udy + vdx}{u_i R_i} \tag{1}$$

This integral is calculated along the planes, which are parallel to the ground surface. Here x-coordinate is parallel to the inlet axis, y-coordinate is perpendicular to it; u and v are respective components of the velocity vector. The integration area is a circle, the center of which is near the vortex center. The circle radius value is chosen such that Γ remains practically unchangeable with increase in radius.

The values of Γ versus non-dimensional height above ground surface z/R_i are presented in fig. 5. The values of Γ remain practically independent of height z/R_i, if it is small, and decrease with increase in inlet height above the ground surface.

A number of calculations were devoted to the investigation of influences of different external parameters on the vortex intensity. The values of Γ versus height H/R_i are presented in fig. 6. The exhaust jet has a radical influence on the vortex intensity. In case H/R_i=1.9 with a jet, the intensity is the same as it is at H/R_i=3.5 without a jet.

An important conclusion can be made from the analysis of calculations for distribution of the vertical component of velocity: the rule of its dependence on height has a universal character (to some degree of accuracy). The maximum values of vertical velocity V_z/U versus height z/R_i are presented in fig. 7. Here U is a value of induced velocity for the given value of H/R_i. Apparently, this rule can only be universal for the near-surface case.

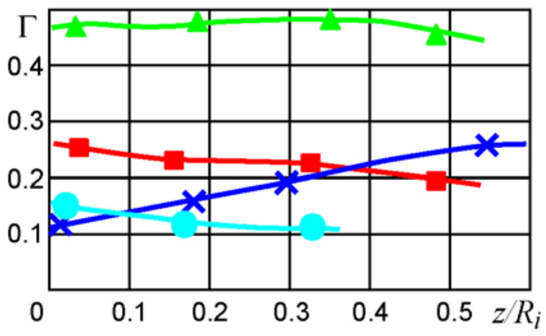

Fig. 5

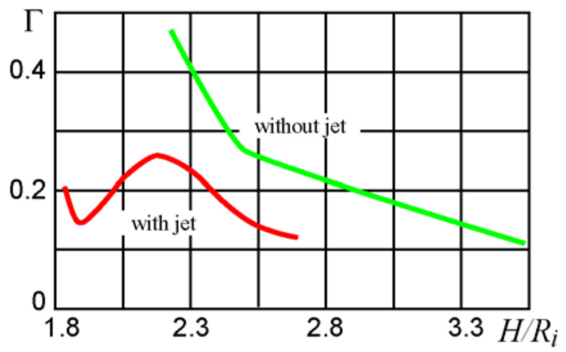

Fig. 6

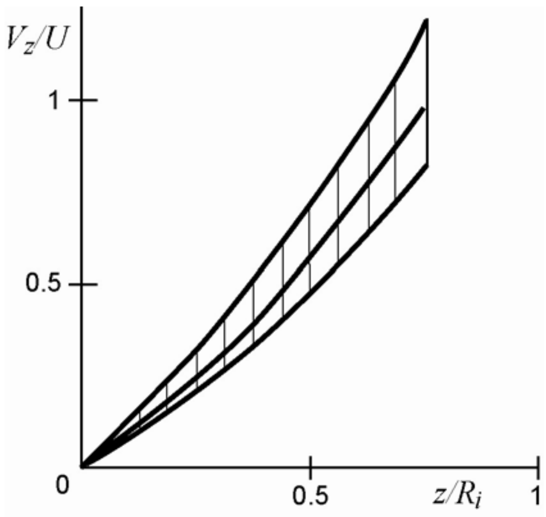

Fig. 7

In accordance with the data in fig.7, the value of the vertical component of the velocity vector V_z achieves the approximate values of U at significant height $z/R_i = 0.6 \div 0.7$ for various inlet heights above ground and in various external conditions.

Calculations data demonstrate that for low z:

$$V_z \sim U\, z/R_i \qquad (2)$$

Pressure distributions in a vortex-type flow were determined from the calculations. The calculations demonstrated that values of pressure gradients are small and that maximum values of pressure difference in the vortex are commensurable with the dynamic pressure determined according to the typical local velocity.

Particles ingestion

The flow ascending from the ground surface transports particles to the air intake. As experience shows, the size of particles can be large enough and achieve up to 10-20 mm and more. In order the flow could lift the particle, it is necessary that the particle drag force f exceeded gravitational force P. To estimate the maximum sizes of particles lifted by flow, from the condition of equilibrium P=f we can deduce:

$$d = \frac{3\rho}{4\rho_P g} \cdot u^2$$

Here u denotes flow velocity, ρ_p stands for particle density, ρ is taken for density of moving medium (it is assumed that $C_x = 1$).

The calculation results stated above show that the velocity values near the ground are not large and do not exceed the value of induced velocity U. And the values of the vertical velocity component varying from zero reach the value $V_z \approx U$ at height $z \approx R_i$. This allows draw a conclusion that big particles are lifted due to accumulation of small particles in the near-ground vortex region, where they create the moving medium with density ρ, which is much greater than air density.

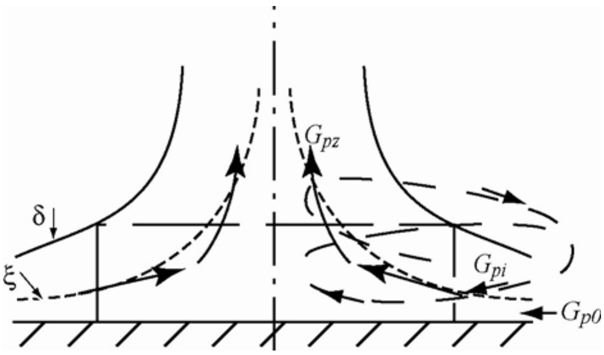

Fig. 8

The vortex flow pattern is illustrated in fig. 8. The particle flux G_{p0} in the near-ground layer ξ moves towards the vortex. The particles inside the vortex scatter and generate an additional flux G_{pi}, which enters the vortex again. As a result, the enlarged flux of particles $G_{p0}+G_{pi}$ enters the vortex. This process repeats until the medium density in the vortex region takes the value ρ_∞, at which the upflow can move to the inlet such number of particles that transports to the vortex from environment. In this case the vertical stream of particles G_{pz} will be

$$G_{pz}=G_{p0}$$

The stationary process is set in this case where the cumulative flux of particles entering the vortex region is equal to G_{pe}. This process can be described by balance equation:

$$G_{pz}=G_{p0}=G_{pe}-G_{pe}K_q,$$

where K_q – particles ejection (separation) coefficient in the vortex flow. (This process is similar to the process that occurs in centrifugal dust-trapping units [5]).

G_{pe} value defines the process of particle accumulation in the vortex. It can be shown that increase in density in the vortex is expressed as

$$\rho_\infty / \rho_o \cong G_{pe}/G_{p0}$$

Here ρ_0 denotes density of the medium, which moves to the vortex in layer ξ. The estimations show that this density is higher than air density by 10-20%.

By analogy with the processes, which take place in centrifugal dust-trapping units [5], it can be shown that the separation coefficient in the vortex is

$$1/(1-K_q)\sim\Gamma,$$

where Γ is a dimensionless vortex intensity determined above (1). So, it follows from this that

$$\rho_\infty \sim \rho_o \cdot\Gamma$$

The flow calculations, with the help of which the velocity distributions in the near-ground layer were determined, enable to estimate the value of the vertical velocity of the medium in the vortex causing big particles lifting.

In accordance with (2), the value of vertical velocity V_z is proportional to height z above the ground surface. The thin layer of particles (dust), initially existing on the surface and having thickness ξ when moving to the vortex center, at first keeps this thickness as the velocity of motion V_r to the vortex center increases and the flow cross-section decreases. But in the region, where V_r begins to decrease, the thickness of particles layer starts to increase according to the rule $h_\xi\sim\xi/(r/R_i)$. On the other hand, $V_z\sim Uh_\xi/R_i$. Therefore

$$V_z=\text{const}\ (\xi/R_i)\ u_i \qquad (3)$$

From the equation for the maximum size of particles being lifted, we obtain that

$$d\sim\bar{\rho}\,V_z^2,\quad \bar{\rho}=\rho_\infty/\rho_p.$$

Because of $\bar{\rho}\sim\Gamma$, i.e. the maximum size of particles being lifted is proportional to the vortex intensity Γ, the values of which and dependence on height H and other parameters were determined in flow calculations. The size of particles is also proportional to the square of velocity V_z determined by (3). V_z is proportional to u_i and to the relative thickness of the initial particle layer ξ on ground. The airflow set this layer in motion. (It can be noted that the ejection effect of the jet causes the decrease in parameter ξ).

The calculation data on the flow dynamics and typical vortex size are needed for the analysis of particle accumulation process in the vortex and determination of typical accumulation time τ.

$$\tau \approx 5 \div 10\ l/U,$$

where l is a typical vortex size.

Conclusion

The calculations of the flow pattern during the air intake operation near the ground surface were made. They adequately enough describe the airflow, vortex initiation as well as flow pattern change in various external conditions: with or without the exhaust jet, at different inlet velocities and variations in the inlet relative height above the ground surface. The flow calculations for various relative inlet heights above ground H/R_i have shown that airflow velocity level near the ground surface corresponds to the values of induced velocities U. This is in agreement with the experimental data of [1]. The analysis of particles transportation from the ground surface to the air intake by vortex flow has demonstrated that sizes of particles are proportional to the vortex intensity Γ and to the square of relative thickness of a thin layer of particles (dust) ξ/R_i that can move along the surface. The action of the engine exhaust jet causes the decrease in this thickness and depends on external conditions.

This research was partly supported by the Russian Fund for Basic Researches, projects No. 02-01-00422 and 05-01-00846, and the Government Program for the Support of the Russian Leading Scientific Schools (NSh-2124-2003-1).

References

[1] J.L. Colehour, B.W. Farquhar. Inlet vortex. J. Aircraft Vol. 8, No. 1, 1971. Pp 39-43.

[2] A.N. Gulyaev, V.E. Kozlov, A.N. Sekundov. A Universal one-equation model for turbulent viscosity. Translated from Izvestiya Rossiiskoi Akademii Nauk, Mechanika Zhidkosti I Gaza, № 4, pp 69-81, 1983.

[3] S. Godunov, A. Zabrodin, M. Ivanov. Numerical solution of multi-dimensional gasdynamics problems. Nauka, Moscow, 1976.

[4] S.Yu. Krasheninnikov, E.G. Rogalskaya. Spreading of turbulent jet in space and near surface. Izvestiya Rossiiskoi Akademii Nauk, Mechanika Zhidkosti I Gaza, 1979, №4. 10 p.

[5] A.K. Gupta, D.E. Lilley, N. Syred. Swirl Flows, Abacus Press, 1984