UDC 503.3.054:629.735.03 (045)

Associate professor K.Synylo, PhD-student A.Krupko
(National Aviation University)

Numerical simulation of the jet from the aircraft engine by OPENFOAM

The work was aimed to simulate the jet from aircraft engine near the ground surface. Numerical investigation of properties and structure of aircraft engine jets allow to get a deep understanding of the fluid mechanics, the jet dynamics and transport process of contaminates and their dilution by jet near the ground.

Aircraft in operation (during approach, landing, taxiing, take-off and initial climb – landing-takeoff cycle or LTO-cycle) and maintenance (aircraft engine run-ups) is a dominant source of impact on LAQ in vicinity of the airport in most cases under consideration [1,2].

The most important feature of the source of emission under consideration is a presence of exhaust gases jet, which contains significant momentum and thermal buoyancy and in accordance may transport contaminant on rather large distances and the rise the plume centerline over the height of engine installation and over the ground surface appropriately. The value of such a distance is defined by engine power setting and installation parameters, mode of an airplane movement, meteorological parameters. The results of the jet model calculations, depending on listed initial data, show that the extent of transport of the jet plumes from aircraft engines may change within the 20...1000 m and sometimes even more [3].

Numerical simulation of the jet from the aircraft engine exhaust was performed in OpenFoam. The design parameters character for engine NK-8-2U of the aircraft Tupolev-154 was used for the research task. Nozzle diameter of the aircraft engine exhaust D = 1.0 m, the height of engine installation h_{EN} above the ground – 3.5 m.

Computational grid

For these tasks a three-dimensional model of a jet was generated in OpenFoam on the basis of the computational grid (fig.1), which was built in a special software program "Gmsh, version 3.0.6.

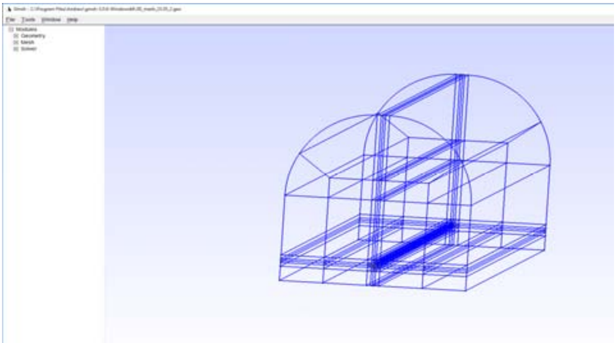
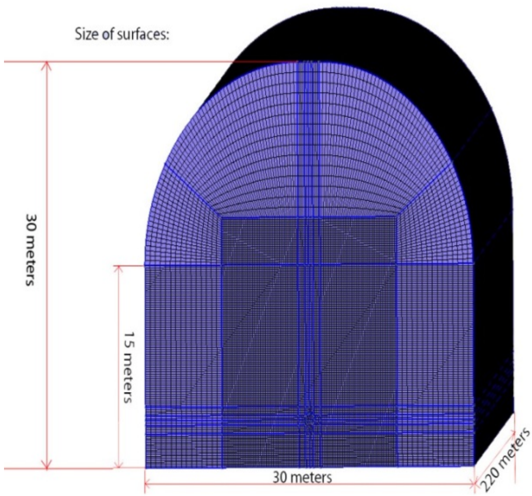


Fig.1. Visualization of geometry model

The size of computational grid is 30*30*220 meters. Total number of cells is 3468084. The most part of computational mesh is represented by the cells of hexahedral shape, the top part – by the cells of the curved shape. This configuration was selected to simplify the problem and optimize the mesh distribution where it is needed mostly (i.e. near the engine exhaust and ground surface) (Fig2).



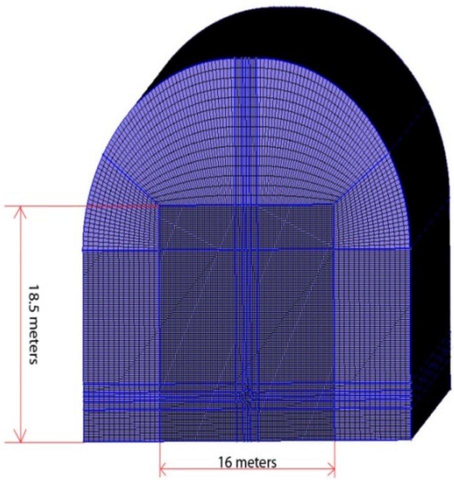


Fig.2. Visualization in vertical plane of computational mesh

The computational domain was divided into few subvolumes to be able to control the mesh precisely (Fig.2). The zone of ground vortices formation – between ground surface and aircraft engine exhaust nozzle – is characterized by structured mesh with higher resolution, with aim to investigate the ground vortices generation processes and basic mechanisms of boundary layer formation, ground surface impact on fluid flow mechanics and particularly Coanda effect occurrence. Zone of engine nozzle exhaust is discretized using a very fine structured mesh to capture the jet development pattern and its vortices structure.

Boundary conditions, turbulence model, solver

For considered task the following boundary conditions were specified to the boundaries of the computational domain of jet flow field, fig.3:

• The nozzle section of aircraft engine exhaust at which jet hot gases enter to computational domain is set as a "velocity inlet" with velocity magnitude 100 m/s (low idle mode of engine operation);

• The computational surfaces adjacent to the engine section at which ambient conditions (wind velocity, wind direction and temperature) enter to computational domain is also set as "velocity inlet" with wind velocity 2 $m \cdot s^{-1}$ and wind direction 0° (normal to boundary). The co-flow is across the domain;

• The external lateral surfaces of computational domain at which ambient conditions (wind velocity, wind direction and temperature) are set as "velocity inlet" also: wind direction and velocity were defined by velocity specification method, X-component of flow direction = 1;

• The ground surface, which is corresponding to the bottom of the computational domain, is set as “wall” implying a non-slip condition for velocity.

• The computational surface opposite to the aircraft engine exhaust nozzle, at which flow field (mixture jet and ambient air) leaves computational volume, was set as “pressure-outlet”.

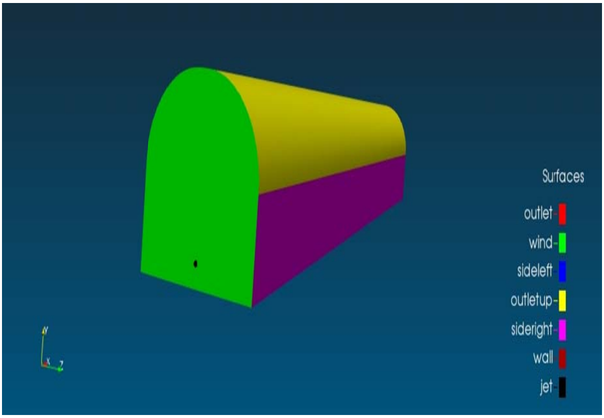


Fig.3. Boundary conditions for simulations of exhaust gases jet from aircraft engine near ground

All calculations were made with **steady-state solver** for incompressible flows (*SimpleFoam*). K-epsilon turbulence model was used to evaluate the turbulence characteristics of fluid flow. The result has converged after 1879 iterations.

All calculations were made with steady-state solver for incompressible flows (SimpleFoam). K-epsilon turbulence model was used to evaluate the turbulence characteristics of fluid flow. The result has converged after 1879 iterations.

The modeling results were proceeded by ParaView, with aim to investigate the fluid mechanisms aspects of the exhaust gases jet from aircraft engine exhaust near the ground surface.

The fig.4 demonstrates the basic mechanisms of the jet behavior, as Coanda and buoyancy effect. So, the exhaust gases jet clings to the aerodrome’s surface for large distance before buoyancy takes over and causes the jet lift and rise above the ground.

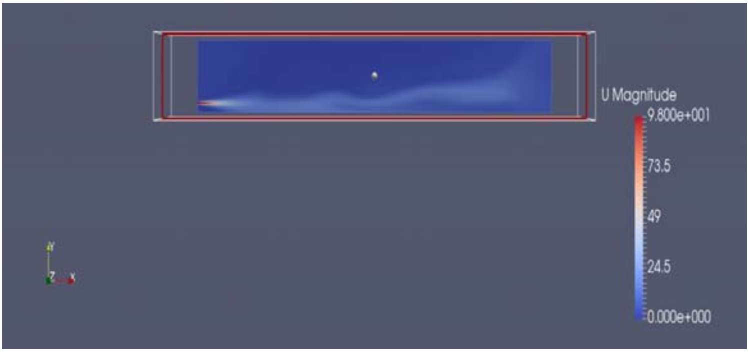


Fig.4. Velocity contours in steamwise direction

The built axial velocity profile (fig.5) and the axis jet (fig.6) allow assess the basic parameters of the jet:

1. height due to the buoyancy effect, ΔH_A = 18.5 m;
2. longitudinal coordinate of the jet axis due to the buoyancy effect, X_A= 130 m;
3. length of jet penetration, S_j=140 m.

The listed basic parameters of the jet (height and longitudinal coordinate due to the buoyancy effect, length of jet penetration) are needed as input data to dispersion modeling of aircraft sources. Exclusion of the fluid mechanisms in considered modeling systems may overestimate the height of buoyancy exhaust gases jet from aircraft engine, underestimate its length and radius of expansion, dispersion characteristics and contaminants concentration values.

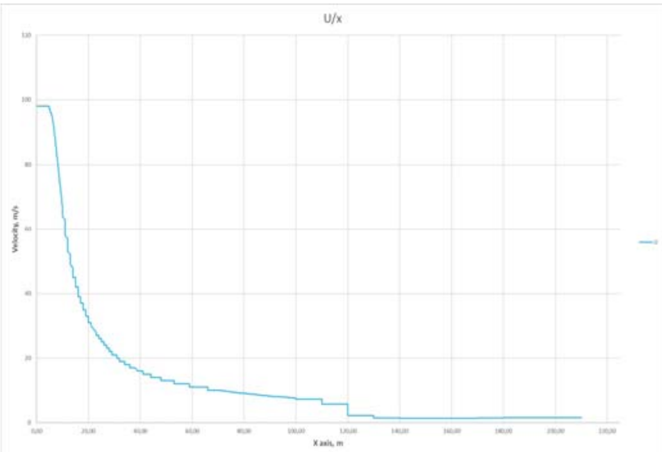


Fig. Maximum velocity decay along the axis

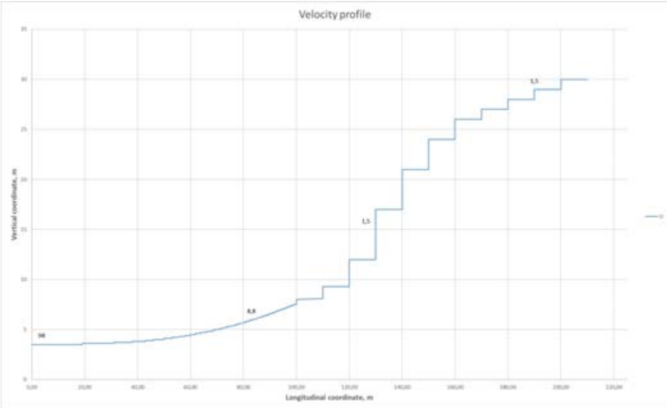


Fig. Buoyancy effect of the jet

Conclusions:

CFD numerical simulations of aircraft engine exhaust jet near to ground surface show that structure, properties and fluid mechanics of jets are influenced by the ground surfaces, providing longer penetration, less rise and appropriate dispersion parameters of the jets, and according little bit higher concentrations of air pollution. So, using obtained results from CFD simulations of aircraft engine jet dynamics allow to improve LAQ modeling systems.

References

1. Zaporozhets O., Synylo K. PolEmiCa – tool for air pollution and aircraft engine emission assessment in airports // 2-nd World Congress Proc.: “Aviation in XXI Century”, Environment Protection Symposium**. –** 2005. – P.4.22 –4.28
2. Abramovich G.I. The theory of turbulent jets - Moscow: Physmatgiz, 1960. - 716 p.
3. Zaporozhets O.I. Outcomes of research of unfavorable conditions of air contamination by effluents of harmful substances from aircraft engines // Problems of environment protection in civil aircraft production. - Kyiv: KIECA, 1989. - P. 13-17.
4. OpenFoam. User Guide,2018:https://openfoam.org/