UDC 503.3.054:629.735.03 (045)
DOI 10.18372/2306-1472.80.14276

Kateryna Synylo[1]
Andrii Krupko[2]
Oleksandr Zaporozhets[3]

NUMERICAL COMPUTATIONS OF EXHAUST GASES JET FROM AIRCRAFT ENGINE UNDER IDLE OPERATIONAL CONDITIONS

[1,2,3]National Aviation University, 1, Lubomyr Husar ave., Kyiv, 03058, Ukraine
E-mails: [1]synyka@gmail.com, [2]kranig15@gmail.com, [3]zap@nau.edu

Abstract

***Purpose**: Currently models for airport air quality are based on the semi-empirical approaches for description of fluid dynamic of exhaust gases jet from aircraft engine and do not take into account an influence of the ground on jet behaviour and the interaction between the jet and the wing trailing vortex system. Eliminating the fluid mechanisms of aircraft wake vortex in models of airport air quality 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. Evaluation the entrainment and mixing processes of the engine emissions in the plume by using CFD-code is an actual task for airport air quality studies. **Methods**: Numerical investigation of properties and structure of aircraft engine jets with CFD codes (OpenFOAM) will give a realistic checked material, on the base of which a necessary scientific reasoning of transportation of the contaminants by engine jets. **Results:** Comparison between OpenFOAM numerical results and semi-empirical jet model calculations (used by complex model PolEmiCa) show that buoyancy effect parameters of exhaust gases decrease twice for wall jet. And the difference between appropriate longitudinal coordinate of buoyancy effect is near to 30%. **Discussion:** using CFD tool allows to improve an airport air quality analysis by providing more objective and accurate input data for further dispersion modelling.*

Keywords: aircraft engine; exhaust gases jet; air pollution; airport air quality; boundary conditions

1. Introduction

There is a growing concern on the pollution resulting from airport operations because of the expansion of air traffic over the years. It is forecasted that future air traffic movements will increase at a mean annual rate of 5 to 7% [1]. According to Schafer et al., 15,000 aircraft are already populating the sky and an expected 2,200 billion of passenger kilometres are flown each year [2].

Although airports provide several benefits for our society, communities in the vicinity of airports are subjected to the deterioration of air quality. This environmental problem is becoming greater because of increasing air traffic, which is quite often associated with the expansion of airports and the continuous growth of moving closer and closer to airports. The analysis of emission inventories from major European and Ukrainian airports has highlighted, that aircraft are the dominant source of air pollution in most cases considered. Aircraft is a special source of air pollution due to some features [16].

The first 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 to 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 [6].

Second, the most part of LTO-cycle the aircraft is maneuvering on the ground, it is subjected to fluid flow that can create **a strong vortex between the ground and engine nozzl**e, which have essential influence on structure and basic mechanisms of exhaust gases jet.

Currently models (PolEmiCa [16, 17], EDMS [18], LASPORT [19]) for airport air quality are based on the semi-empirical approaches for description of fluid dynamic of exhaust gases jet from aircraft engine and do not take into account an influence of the ground on jet behaviour and the interaction between the jet and the wing trailing

http://www.nau.edu.ua

vortex system. Eliminating the fluid mechanisms of aircraft wake vortex in considered modelling 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.

Evaluation the entrainment and mixing processes of the engine emissions in the plume by using CFD-code is an actual task for airport air quality studies.

2. Aim of work

Improvement of complex model PolEmiCa, particularly assessment parameters of exhaust gases jet (height and longitudinal coordinate of buoyancy effect, length of jet penetration) by using CFD codes.

3. Review and analysis of the studies

Over the last few decades, several analytical and numerical studies [5 - 10] have been devoted to the problem of the wake vortex evolution near the ground. Chan [5] highlighted the importance of the vortex circulation effect on pollutant dispersion and concluded the variation of pollution concentration distribution in the control volume is primary due to the circulation of vortices near the ground. Harvey&Perry [6] examined the case of vortex pair approaching a no-slip wall in a viscous fluid by test in wind tunnel. On the basis of analysis of flight test data and wind tunnel tests, it was observed, that the vortices approach the ground, induce a cross flow and create a boundary layer. An adverse pressure gradient causes the boundary layer to separate from the ground and roll-up into oppositely signed vortex, which causes the primary vortex to rebound from the ground [6]. Atias &Weihms investigated the motion of a vortex pair near the ground numerically using a discrete vortex method [7]. It was found, that the vortices near the ground rebound and follow upward, which is explained by the formation of a boundary layer at the ground [7]. Unsteady numerical simulation of the interaction of vortices with the ground was conducted by Spalarat et al. [8] by using different turbulent models (e.g. Spalart-Almaras and k-ώ).

This simulation results in considerable overestimation of vortex dissipation and thus inaccurate ground vortices behavior. On the basis of a more appropriate model (Spalart-Schur) [8] it was concluded that vortex rebound height is mainly determined by the parameters of vorticity of boundary layer near the ground.

The wake vortices are considered to be “in ground effect” (IGE) when located within one initial separation of the ground (i.e., z < bo). The proximity of the ground and wind have a strong effect on the development. The ground mainly affects the vortices trajectory and leads to the formation of secondary vortices coming from the boundary layer. The path of the descending vortices gradually diverges when they reach the ground [5 - 9]. This is an in viscid phenomenon and it can be explained by the fact the ground acts like a symmetry boundary. The viscous effects produce a boundary layer separation which may cause the creation of a secondary vortex. The interaction between the secondary vortex and the primary vortex changes its trajectory and results in vortex rebound [5, 9].

Ash et al. (1994) also conducted numerical simulation of aircraft wake vortex transport near the ground by using a Reynolds turbulence model to examine cross wind and atmospheric turbulence impact on ground effects. It was reported, that the ambient variables may have an essential influence on wake vortex behavior in terms of lifetime and hazard [10 - 13].

Turbulent plane free and wall jet simulations were carried out by S.S. Aloysius and L.C. Wrobel [9 - 11] to validate the computational models and to assess the impact of the presence of the solid boundary on the fluid flow and its dispersion properties.

This review has demonstrated the need of applicability of CFD methods to airport-related flow and dispersion problems, and their capability to characterize source dynamics. The results can help dispersion modelers with better aircraft plume dynamics representation and benefit the airport air quality regulation.

4. Numerical simulation of the jet from aircraft engine NK-8-2U via OpenFOAM

OpenFOAM is the free, open source CFD software released and developed primarily by OpenCFD Ltd since 2004. It has a large user base across most areas of engineering and science, from both commercial and academic organisations. OpenFOAM has an extensive range of features to solve anything from complex fluid flows involving chemical reactions, turbulence and heat transfer, to acoustics, solid mechanics and electromagnetics [14].

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.

4.1. 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. Gmsh is a free 3D finite element mesh generator with a built-in CAD engine and post-processor. Its design goal is to provide a fast, light and user-friendly meshing tool with parametric input and advanced visualization capabilities. Gmsh is built around four modules: geometry and user-friendly meshing tool with parametric input and advanced visualization capabilities. Gmsh is built around four modules: geometry mesh, solver and post-processing.

The specification of any input to these modules is done either interactively using the graphical user interface, in ASCII text files using Gmsh's own scripting language (.geofiles), or using the C++, C, Python or Julia API [15].

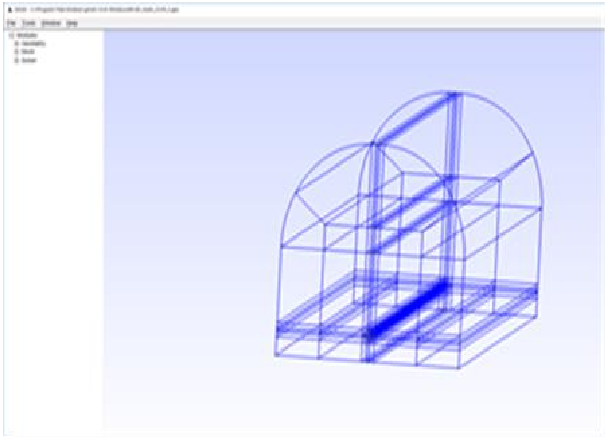

Fig.1. Visualization of geometry model

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) (Fig.2).

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.

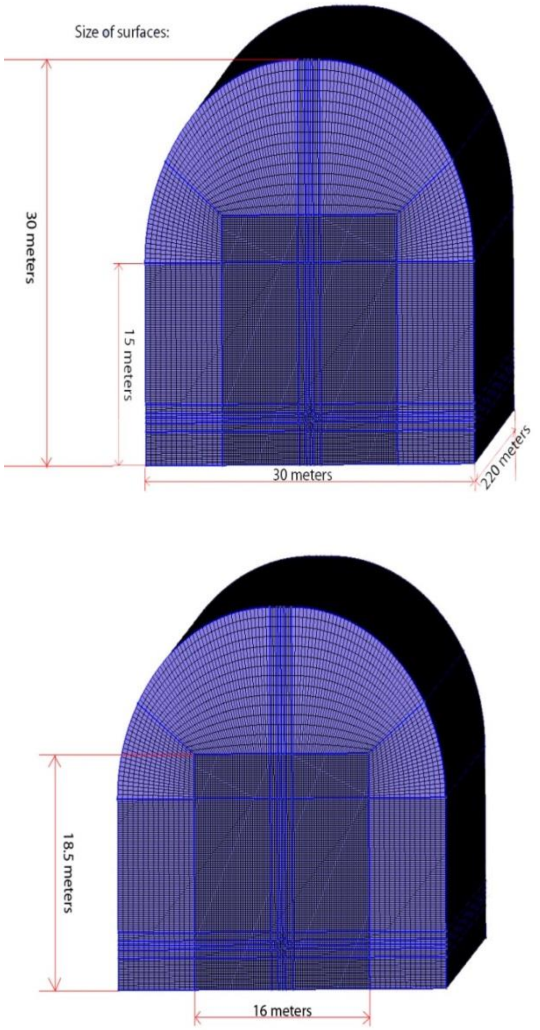

Fig.2. Visualization in vertical plane of computational mesh

3.2 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·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”.

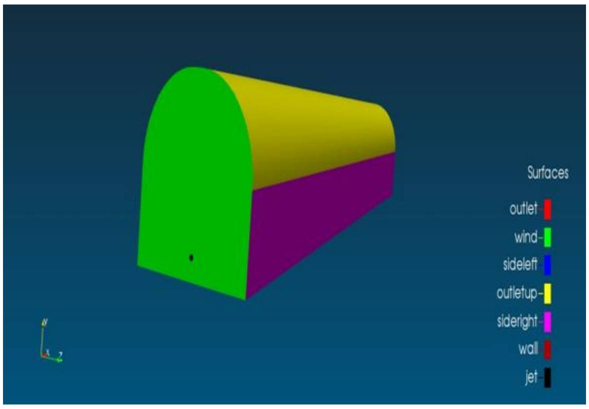

Fig.3. Boundary conditions for simulations of exhaust gases jet from aircraft engine near ground

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 1911 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 1911 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.

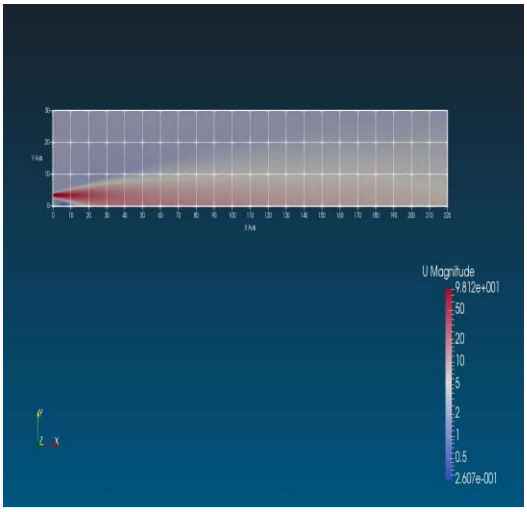

Fig.4. Velocity contours in steamwise direction due to RANS modeling

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.

All calculations were also made with steady-state solver for incompressible flows (*buoyantBoussinesqSimpleFoam*). K-epsilon turbulence model was used to evaluate the turbulence characteristics of fluid flow.

Obtained results demonstrate the basic mechanisms of the jet behaviour, 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.4-8).

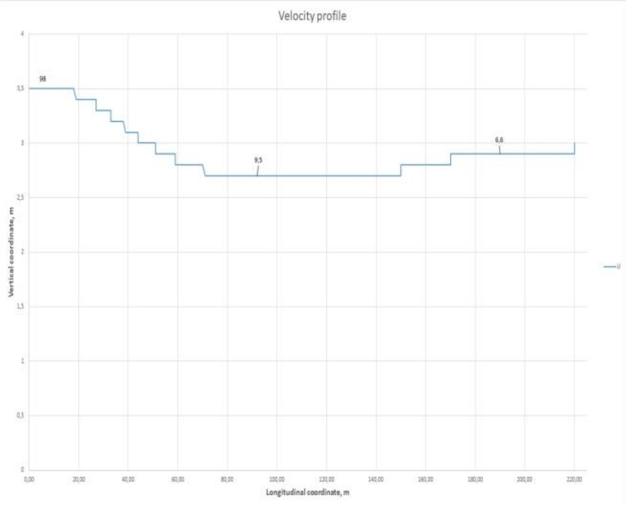

Fig.5. Velocity profile in steamwise and vertical direction due to RANS modeling

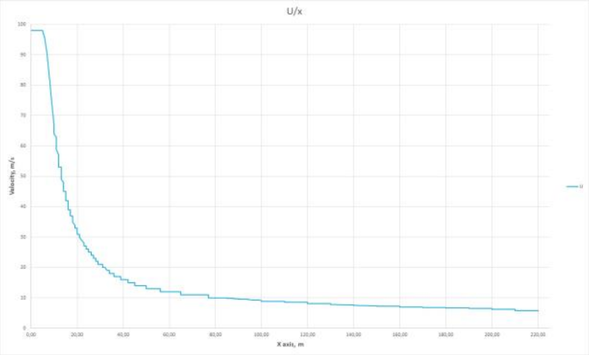

Fig.6. Changing of velocity depending on distance due to RANS modeling

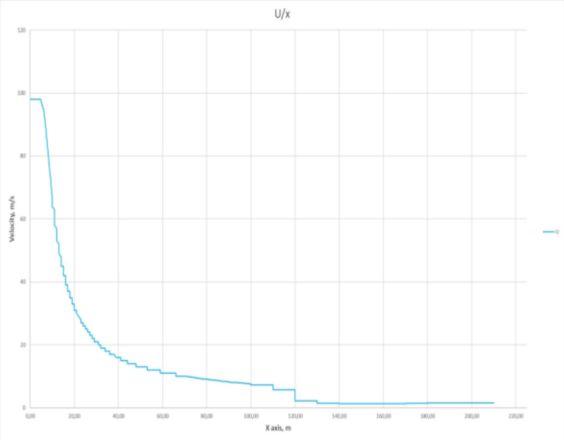

Fig. 7. Velocity profile of the exhaust gases jet from aircraft engine NK-8-2U

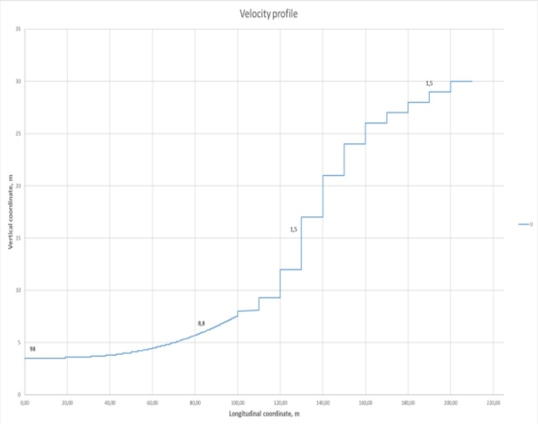

Fig. 8. Axis of the exhaust gases jet from aircraft engine NK-8-2U due to velocity gradient

The obtained longitudinal coordinate and height of buoyancy effect due to simulation by Open FOAM were compared with previous modeling by FLUENT6.3 and PolEmiCa for the real NK-8-2U engine under idle operation conditions (U_0= 98 m·s^{-1}, T_0 = 423 K) and same ambient conditions (U_w = 2 m·s^{-1}, ψ = 0°, T_A = 300 K), Table 1.

Table 1

Comparison of calculated parameters of the jet from NK-8-2U engine

Parameters	Calculation results of model			
Initial conditions U_j= 98 m·s^{-1}, T_j = 423K	Open FOAM	Fluent 6.3 Mesh 1	Fluent 6.3 Mesh 2	PolEmiCa previous version
Height of jet rise Δh_A, m	12,0	20,5	21,0	40,4
Longitudinal coordinate of jet rise X_A, m	125,0	170,0	165,0	125,0

Comparison between OpenFOAM numerical results and semi-empirical jet model calculations (used by complex model PolEmiCa) show that buoyancy effect parameters of exhaust gases decrease twice for wall jet, tab. 1.

And the difference between appropriate longitudinal coordinate of buoyancy effect is near to 30%.

The found difference is explained by absence of the account of ground surface impact on jet structure in complex model PolEmiCa that may lead to overestimation of the height (near to twice!) and underestimation of longitudinal coordinate of buoyancy effect.

4. Conclusions

On the ground of obtained CFD results for basic jet parameters (height and longitudinal coordinate of jet axis arise due to buoyancy effect, length of jet penetration) and their comparison with semi-empirical jet model used in PolEmiCa it may be concluded, that using CFD tool allows to improve an airport air quality analysis by providing more objective and accurate input data for further dispersion modeling.

References

[1] Graham and D. Raper. “Air Quality in Airport Approaches: Impact of Emissions entrained by Vortices in Aircraft Wakes”. Available at: http://www.cate.mmu.ac.uk/documents/Publications/Woct03.pdf. (2003).

[2] K. Schäfer, C. Jahn, P. Sturm, B. Lechner and M. Bacher. “Aircraft Emission Measurements by Remote Sensing Methodologies at Airports”. Atmospheric Environment, Vol. 37, pp. 5261-5271. (2003).

[3] Chan T.L., Dong G., Leung C.W., Cheung C.S., Hung W.T. Validation of a 2D Pollutant Dispersion Model in an Isolated Street Canyon. – Atmospheric Environment. – 2002. – №36. – P.861-872.

[4] Harvey, J. K. & Perry, F. J. Flowfield produced by trailing vortices in the vicinity of the ground. – AIAA J. – 1971. – №9 (8) – P.1659-1660.

[5] Atias, M., Weihs, D. Motion of aircraft trailing vortices near the ground. – J. Aircraft. – 1984. – №21(10). – P.783-786.

[6] Spalart, P. R., Strelets, M. K., Tra vin, A. K., & Shur, M. L. Modeling the Interaction of a Vortex Pair with the Ground. – Fluid Dynamic. – 2001. – № 36(6). – P.899-908.

[7] Aloysius Syoginus S., Wrobel Luiz C. ALAQS–CFD Comparison of Buoyant Free and Wall Turbulent Jets// School of Engineering & Design Brunel University. – UK, 2007. – 58 p.

[8] Aloysius Syoginus S., Wrobel Luiz C., Pearce D. ALAQS–Comparison of CFD and Lagrangian Dispersion Methods – Simple Scenario during Take-off // School of Engineering & Design Brunel University. – UK, 2007. – 32 p.

[9] Syoginus S. Aloysius and Luiz C. Wrobel On the Use of Near Field Computational Fluid Dynamics for Improving Airport Related Dispersion Models // Third International Conference On Research In Air Transportation Fairfax, Va, June 1-4 2008, 8 p.

[10] M.R. Davis and H. Winarto. “Jet Diffusion from a Circular Nozzle above a Solid Plane”. Journal of Fluid Mechanics. Vol. 101, part 1, pp.201-222. (1980).

[11] Bradbury L.J.S. The Structure of a Self-Preserving Turbulent Jet. – J. Fluid Mech. – 1965. – № 23 – P.31-64.

[12] Forthmann E. Turbulent Jet Expansion. English translation, NACA Technical Memorandum TM-789. – 1936.

[13] Wygnanski, I., Katz, Y., Horev, E. On the Applicability of Various Scaling Laws to the Turbulent Wall Jet”. – J. Fluid Mech. – 1992. – Vol. 234 – P. 669-690.

[14] Wayson R.L., Flemming G.G., Eberhard W., Kim B. Final report: The use of LIDAR to characterize aircraft initial plume characteristics // FAA-AEE-04-01. – 2004. – 35 p.

[15] User Manual of Open Foam. Available at: https://www.openfoam.com/

[16] Synylo K., Zaporozhets O., FröhlichJ., Stiller J. Improvement of Airport Local Air Quality Modeling // Journal of Aircraft. ‑ 2017. ‑ Vol. 54. ‑ №. 5. ‑ P. 1750 ‑1759 (Scopus).

[17] 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

[18] Emissions and Dispersion Modelling System (EDMS) // Reference Manual. FAA-AEE-01-01. U.S. Department of Transportation Federal Aviation Administration, Washington, D.C. CSSI, Inc., Washington, D.C. ‑ September 2002.

[19] Janicke Consulting, LASPORT version 1.3 Reference Book. ‑ November 2005. ‑ 93 p.

К.В. Синило[1], А.І. Крупко[2], О.І. Запорожець[3]

Чисельне моделювання струменя газів від авіадвигуна за умов малого газу

[1,2,3]Національний авіаційний університет, просп. Любомира Гузара 1, Київ, 03058, Україна
E-mails: [1]synyka@gmail.com, [2]kranig15@gmail.com, [3]zap@nau.edu

Мета: У даний час моделі якості повітря в аеропорту базуються на напівемпіричних підходах для опису динаміки струменя вихлопних газів авіадвигуна і не враховують вплив поверхні аеродрому на поведінку струменя та процеси взаємодії струменя з вихорами від крила. Виключення зазначених механізмів у моделях якості повітря в аеропорту може завищити висоту плавучості струменя вихлопних газів від авіадвигуна, не об’єктивно обчислити далекобійність струменя та радіус розширення, дисперсії та відповідно величини концентрацій. Оцінка процесів переносу та розбавлення домішок струменем газів від авіадвигуна за допомогою CFD-коду є актуальним завданням для дослідження якості повітря в аеропорту. **Методи**: Чисельне дослідження властивостей та структури струменя газів від авіадвигуна за допомогою CFD кодів (OpenFOAM) надасть реалістичний перевірений матеріал, що забезпечить необхідне наукове обгрунтування оцінки рівнів забруднення внаслідок викидів авіадвигунів. **Результати**: Порівняння чисельних результатів OpenFOAM та розрахунків напівемпіричної реактивної моделі (використовувана складною моделлю PolEmiCa) показує, що параметри впливу плавучості відпрацьованих газів зменшуються вдвічі для обмеженого струменя. А різниця між відповідною поздовжньою координатою ефекту плавучості становить близько 30%. **Обговорення**: використання інструменту CFD дозволяє вдосконалити аналіз якості аеропорту в аеропорту, надаючи більш об'єктивні та точні вхідні дані для подальшого моделювання дисперсії.

Ключові слова: авіадвигун; струмінь відпрацьованих газів; забруднення атмосферного повітря; місцева якість повітря; граничні умови

К.В.Синило[1], А.И.Крупко[2], О.И.Запорожец[3]

Чисельное моделирование струи газов от авиадвигателя при условиях малого газа

[1,2,3]Национальный авиационный университет, просп. Любомира Гузара 1, Киев, 03058, Украина
E-mails: [1]synyka@gmail.com, [2]kranig15@gmail.com, [3]zap@nau.edu

Цель: В настоящее время модели качества воздуха в аэропорту базируются на полуэмпирических подходах к описанию динамики струи выхлопных газов авиадвигателя и не учитывают влияние поверхности аэродрома на поведение струи и процессы взаимодействия струи с вихрями от крыла. Исключение указанных механизмов в моделях качества воздуха в аэропорту может завысить высоту плавучести струи выхлопных газов от авиадвигателя, необъективно оценить дальнобойность струи и радиус расширения, дисперсии и соответственно величины концентраций. Оценка процессов переноса и разбавления примесей струей газов от авиадвигателя с помощью CFD-кода является актуальной задачей для исследования качества воздуха в аэропорту. **Методы**: Численное исследование свойств и структуры струи газов от авиадвигателя с помощью CFD кодов (OpenFOAM) предоставит реалистичный проверенный материал, который обеспечит необходимое научное обоснование оценки уровней загрязнения вследствие выбросов авиадвигателей. **Результаты**: Сравнение численных результатов OpenFOAM и расчетов полуэмпирической реактивной модели (используемая сложной моделью PolEmiCa) показывает, что параметры влияния плавучести отработанных газов уменьшаются вдвое для ограниченного струи. А разница между соответствующей продольной координатой эффекта плавучести составляет около 30%. **Обсуждение**: использования инструмента CFD позволяет усовершенствовать анализ качества аэропорта в аэропорту, предоставляя более объективные и точные исходные данные для дальнейшего моделирования дисперсии.

Ключевые слова: авиадвигатель; струя отработавших газов; загрязнение атмосферного воздуха; местное качество воздуха; граничные условия

Kateryna Synylo (1979). Candidate of Engineering.
Associate Professor. National Aviation University
Education: Odessa Hydrometeorological institute, 2001
Research area: Methods of local air quality regulation inside the airport
Publications: 57
E-mail: synyka@gmail.com

Andrii Krupko (1990). PhD-student.
National Aviation University
Education: National Aviation University, 2001
Research area: Modeling of airport air pollution
Publications: 7
E-mail: kranig15@gmail.com

Oleksandr Zaporozhets (1956)
D. Sc., Professor.
Education: Kiev Institute of Civil Aviation Engineers, 1978
Research area: Development of Models and Methods of Information Provision for Environment Protection from Civil Aviation Impact
Publications: more than 200 papers in refereed journals, 4- monographs and 6 -patents.
E-mail: zap@nau.edu.ua