



HAL
open science

A study of flow and initial stage of water condensation in the exhaust jet of the aircraft turbofan engine

Maria Anatolievna Lobanova, Yury Michailovich Tsirkunov

► **To cite this version:**

Maria Anatolievna Lobanova, Yury Michailovich Tsirkunov. A study of flow and initial stage of water condensation in the exhaust jet of the aircraft turbofan engine. 2nd ECCOMAS Young Investigators Conference (YIC 2013), Sep 2013, Bordeaux, France. hal-00855869

HAL Id: hal-00855869

<https://hal.science/hal-00855869>

Submitted on 30 Aug 2013

HAL is a multi-disciplinary open access archive for the deposit and dissemination of scientific research documents, whether they are published or not. The documents may come from teaching and research institutions in France or abroad, or from public or private research centers.

L'archive ouverte pluridisciplinaire **HAL**, est destinée au dépôt et à la diffusion de documents scientifiques de niveau recherche, publiés ou non, émanant des établissements d'enseignement et de recherche français ou étrangers, des laboratoires publics ou privés.

A study of flow and initial stage of water condensation in the exhaust jet of the aircraft turbofan engine

M. Lobanova^{*}, Yu. Tsirkunov

^a Baltic State Technical University
190005 Saint-Petersburg, 1-st Krasnoarmeyskaya 1

^{*} lbnv.spb@gmail.com

Abstract. *The paper describes the results of numerical study of flow in the exhaust jet of turbofan engine CFM 56-3. Influence of computational domain decomposition, grid refinement and flow model on the jet flow field is discussed. Special attention is payed to simulation of nucleation and condensation processes in the exhaust jet. Growth of water clusters in the jet and cluster distribution in size are obtained.*

Keywords: turbofan engine; exhaust jet; water vapor condensation; cluster distribution in size; grid refinement.

1 INTRODUCTION

Investigation of physical-chemical transformations in essentially inhomogeneous gas flows imposes high demands on accuracy of fluid flow field calculations that is usually accomplished by grid refinement and its adaption to the flow structure. Physical-chemical transformations often proceed not in the whole flow, but only in a confined flow area. The flow in exhaust jets behind an aircraft provides an example of such a confined area which is a small part of an aircraft jet-vortex wake. Simulation of flow in an exhaust jet and modeling of water condensation in it is the subject of the present study. This problem is a part of the significantly more complex problem of flow simulation around an aircraft and development of two-phase jet-vortex wake behind it [1].

As is known, the water vapor is an inherent component of aviation kerosene combustion products. Interest in the study of the process of water vapor condensation is due to several reasons. Condensation trails (contrails) can result in formation of cirrus clouds that violate the natural Earth's radiative balance and may contribute to climate change [2]. Local and global aviation influence on the atmosphere is the subject of close attention and discussions both among scientists and environmentalists and even among politicians. A wake behind a modern heavy cruise aircraft represents a strongly disturbed vortex flow. Another aircraft can suffer an accident, getting into such a vortex structure. Drops and ice crystals of water, forming in exhaust jets, can be used for visualization of vortex wake and vortex danger degree prediction. Such diagnostics is of great importance in regions with high-density air traffic. The phenomenon of arising and evolution of contrails is the subject of several sciences and their interface, such as aerodynamics, physical-chemical gas dynamics, non-equilibrium thermodynamics, molecular gas dynamics, physics of atmosphere. Therefore, all these disciplines need to be taken into account altogether in order to provide correct physical description of this phenomenon.

This paper presents the results of computational simulation of the flow structure and modeling of the initial stage of water vapor condensation in exhaust jet of the turbofan engine CFM 56-3, which is used in passenger planes Boeing-737. The main aim of the work was to study the influence of computational domain decomposition, grid refinement and the flow model on the jet flow field. Attention was focused on distribution of temperature along and across the jet, which is important for the following calculation of water condensation. A search for grid settings with a relatively small number of cells, providing the acceptable accuracy for calculating turbofan engine jet, is of great importance for full numerical simulation of flow around an aircraft and in the jet-vortex wake, because a required number of grid cells in the full problem is many times larger than that for the single jet.

2 NUMERICAL SIMULATION

2.1 Modeling and numerical approaches

Calculation domain was a cylinder with a diameter of 7.14 m and 20.73 m long, which roughly corresponds to 9 and 27 external diameters of the exhaust duct annular outlet. Diameters of exhaust duct annular outlet are 0.51 and 0.76 m, and diameters of bypass duct annular outlet are 1.07 and 1.43 m.

Jet stream has been modeled on three variants of structured block grids with the number cells 1, 4 and 16 million and on unstructured grid. Spatial discretization of the computational domain has been adapted to the geometry of ducts outlets and to the flow features, in particular, to the gradient of flow parameters at the nozzles exits and in the jet. Several models of flow were used: the Euler, Navier–Stokes and Reynolds equations with Menter $k-\omega$ SST turbulence model. In the latter case, in addition to distributions of temperature and vorticity, field of turbulence kinetic energy was computed. Comparison of results obtained on the basis of the Euler and Navier–Stokes models made it possible to estimate the numerical viscosity of simulation. The following comparison with the results of flow simulation on the basis of the Reynolds equations clarified the role of turbulent viscosity in formation of fields of temperature and other parameters. Temperature and velocity at exits of exhaust and bypass ducts were taken from the thermodynamic calculation of fuel combustion products as boundary conditions. Their values were found to be 883.36 K and 236.4 K, and 517.37 m/s and 366.86 m/s respectively.

For simulation of the initial stage of water vapor condensation in a jet, the discrete condensation model was used [3]. Initially it was assumed that the formation of clusters consisting of two molecules is the result of three particle collisions: two monomers and a third particle, carrying out the energy of recombination. Also further molecules clashes with already formed clusters and inter-clusters collisions were accounted. The theory is based on selection of a number of discrete coupled states of molecules and consideration of kinetics of the transitions between these states. The advantage of this approach is the ability to analyze and simulate the stage preceding the formation of the critical embryo, which, as is well known, the classical theory of condensation does not allow to do.

2.2 Discussion of results

In Figures 1, 2 and 3 the results of calculations for the RANS approach with Menter $k-\omega$ SST turbulence model are presented. The CFD simulation has been done using the ANSYS software (ANSYS ICEM CFD, ANSYS CFX). The shown results are obtained for the case of 4 million grid cells. The z -coordinate in Figures is directed toward the opposite direction of flow and is calculated from the exhaust duct exit. Distribution of water clusters in size is calculated using the software by our colleague Dr. A.A. Gorbunov.

In the Figure 1 on the left hand side are given seven streamlines going from the exhaust duct outlet and on the right hand side are shown graphics of turbulent kinetic energy along these streamlines as a function of distance from the nozzle z . It is seen that lines are different till $z = -10$ m and then they merge until the end of the calculation domain. Turbulence kinetic energy displays a flow turbulence level that in the turbofan jet case has maximum in the area, where exhaust and bypass ducts streams are mixing. Downstream flow becomes more homogeneous.

Figure 2 presents plots of the temperature distribution along the same seven streamlines (left) and a field of temperature in the calculation domain in the plane of symmetry (right). As mentioned, the temperature field in the jet is important for the following calculation of water condensation. Temperature along the streamline #1 that is closer to the flow from the bypass duct falls faster than the temperature along the streamline #7 that goes along the engine geometry surface. That's why temperature along the 7th streamline for the first five meters stays very high and only after $z = -6$ m it starts to fall from 883.36 K to 390 K. Downstream, when the flow becomes more homogeneous, all temperature lines on the plot merge as temperature in the stream tends to a single value.

Interesting results are shown in Figure 3. The left plot demonstrates the growth of the number of water clusters of fixed size in the 4th central streamline. At the exhaust duct outlet existent clusters contain not more than 6 molecules of water. Larger clusters appear with increasing distance from the nozzle at the same time keeping great number of clusters with a few number of water molecules (see also the right plot). The largest cluster at the end of calculation domain contains about 20 molecules H_2O . Larger particles cannot be formed because of high temperature at the outlet boundary (390 K at $z = -20$ m). The right plot in Figure 3 shows the distribution of clusters in size in the taken streamline #4 at different relative distance from the engine exit.

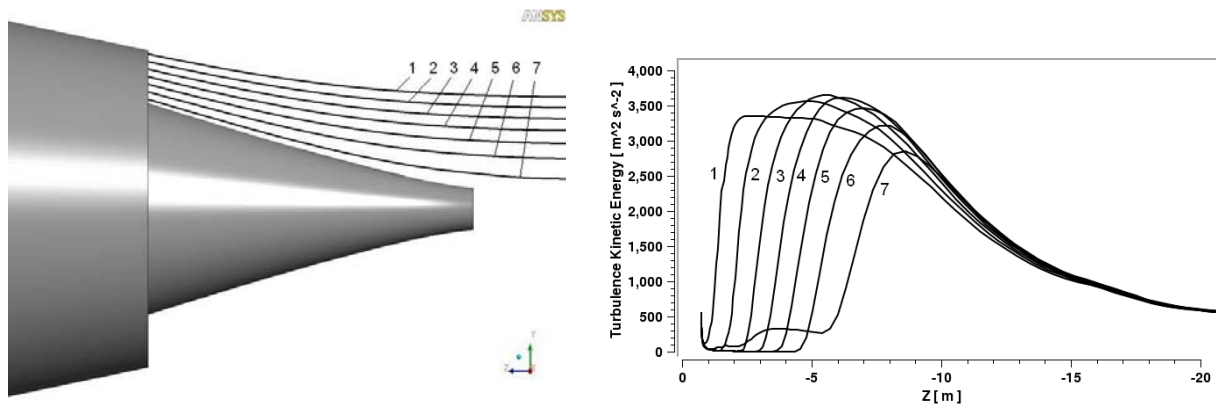


Figure 1: Streamlines going from the exhaust duct outlet (left) and the distribution of turbulence kinetic energy along these streamlines versus distance from the nozzle z (right).

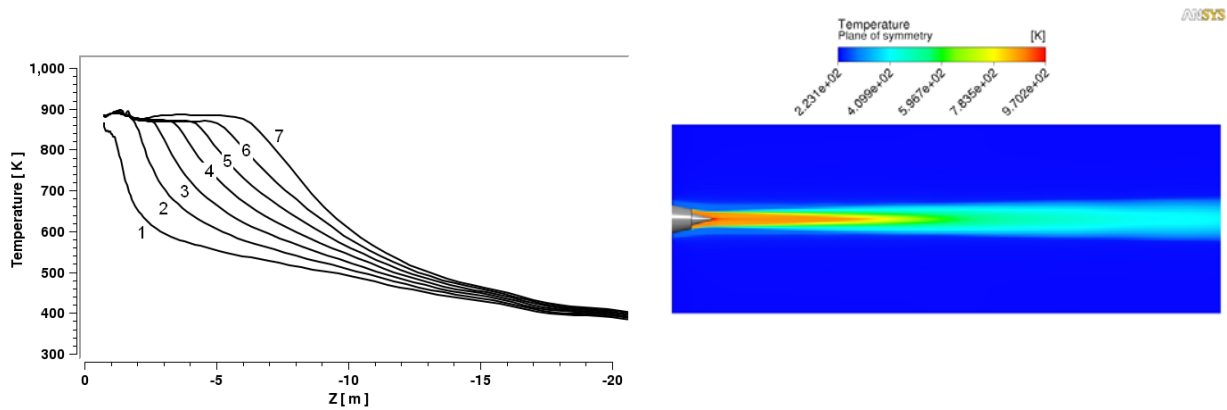


Figure 2: Temperature along seven streamlines from exhaust duct outlet versus distance from the nozzle z (left) and the field of temperature in the calculation domain in the plane of symmetry (right).

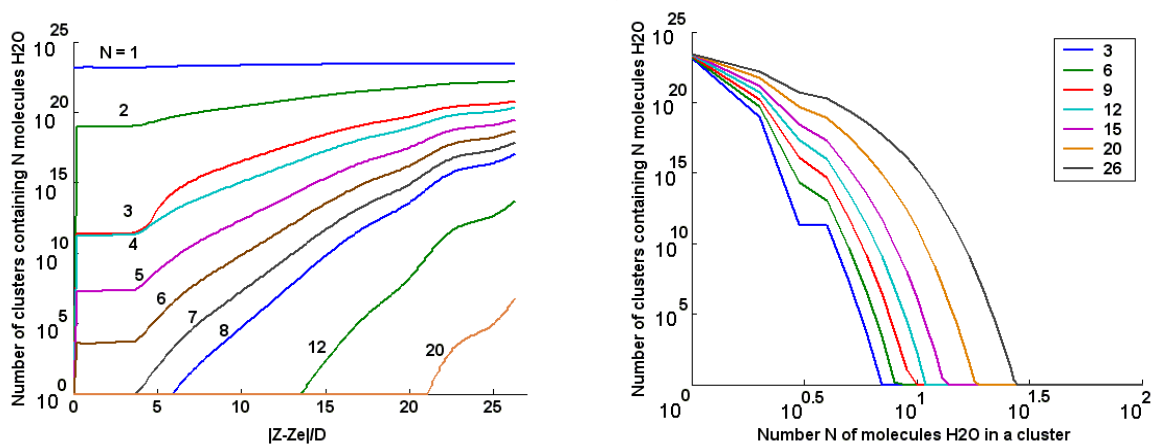


Figure 3: Number of water clusters with N molecules versus relative distance from the exhaust duct outlet (left) and distribution of clusters in the number of molecules at different relative distances from the nozzle exit (right). Z_e is z -coordinate of exhaust duct outlet, and $D = 0.76$ m is its external diameter. All curves are obtained for the central jet streamline #4 (see in Fig. 1).

3 CONCLUSIONS

Presented work is an intermediate stage of solving the more complex problem of flow simulation around an aircraft and development of two-phase jet-vortex wake behind it. The flow structure and the initial stage of water

vapor condensation in exhaust jet of the turbofan engine CFM 56-3 were investigated. Jet stream has been modeled on three variants of structured block grids with the number of cells 1, 4 and 16 million and on unstructured grid. Several models of flow were considered: the Euler, Navier–Stokes and Reynolds equations with Menter $k-\omega$ SST turbulence model. It was found that jet flow properties strongly depend on turbulent viscosity and grid refinement. The study of temperature distribution along and across the exhaust jet has shown that temperature decreases unequal across the jet and temperature value influence on cluster size. The process of water clusters growth in the exhaust jets was simulated, and the cluster size distributions at different distance from the exhaust duct exit was obtained and explained from the physical reasons.

ACKNOWLEDGEMENTS

The authors wish to express their thanks to Dr. A.A. Gorbunov for the help in calculations of the condensation process and to Dr. S.I. Igolkin for discussion of the presented results. This work has been supported by the Russian Foundation for Basic Research through grant No. 12-08-01282.

REFERENCES

- [1] Lobanova, M.A., Tsirkunov, Y.M.. Numerical simulation of a jet-vortex wake behind a cruise aircraft. Paper No. 1823, 14 p. In: *CD-ROM Proceedings of the 6th European Congress on Computational Methods in Applied Sciences and Engineering (ECCOMAS 2012)*, Vienna, Austria, 10–14 September 2012. Eds.: Eberhardsteiner, J.; Böhm, H.J.; Rammerstorfer, F.G., Publisher: Vienna University of Technology, Austria, 2012. ISBN: 978-3-9502481-9-7.
- [2] Intergovernmental Panel of Climate Change (IPCC): *Aviation and the Global Atmosphere*. Cambridge University Press, New-York, 1999.
- [3] Gorbunov, A.A., Igolkin, S.I.. Size distribution of clusters in the mixture of water vapor and nitrogen. *Physics of Aerodispersed Systems* 41:31–39, 2004.