INSTABILITY MECHANISMS OF THE AIRCRAFT WAKE VORTICES

Moldoveanu Cristian-Emil

Military Technical Academy "Ferdinand I"
39-49 Av. George Cosbuc, Bucharest, Romania
Fax: +32(0)2 7376352
E-mail: moldoveanu.cristian@mta.ro

Abstract: *Developing air traffic and introducing large aircrafts in use in the group of transport aircrafts has led to the necessity to optimize separation distances between aircrafts, especially near airports. These distances are imposed by airport safety and security conditions, related to the action of the wake turbulence generated by an aircraft on another. At the edge of the aircraft wings, longitudinal vortices are created by pressure differences inside the boundary layers and rotated in opposite senses. It can constitute a danger to another aircraft that flies in this wake, especially during takeoff and landing. This study presents the modelling methods of the turbulence using Large Eddy Simulation and some applications in the case of wake vortex.*

1. Introduction

Any object moving in the air leaves behind it a more or less organized wake. In the case of an aircraft, wake turbulence is composed of persistent vortices, which constitute a danger for the following aircraft. These vortices are due to the flow of air at the end of the wing. Indeed, the difference in pressure between the bottom and the top tends to cancel at the extremity of the wings. It is this pressure difference that generates lift on each wing and the generated vortex will be even more intense if this lift will be great. As the lift is directly proportional to the weight of the aircraft, the intensity of wake vortex is bound also [3].

Typically, a pair of contrarotatifs vortices is very persistent. It descends into the atmosphere with a vertical speed of approximately 5 m/s (20 km/h). A small aircraft that is experiencing these vortices can be influenced by a violent rolling motion of the air. In the presence of the wind, the atmosphere is generally turbulent. If this turbulence is sufficiently intense, it is capable of destroying wake vortex by causing their misalignment and by promoting their interaction (if two opposed direction of rotation vortices are touching, they will decay). In most cases, the presence of atmospheric disturbances eliminates the danger: aircraft often encounter these phenomenon residues without being able to distinguish them from the ambient atmospheric turbulence. But in a calm atmosphere, wake vortices persist long time. This makes them dangerous [2].

2. Large Eddy Simulation of Turbulent Flow

There is a large number of mathematical methods for the resolution of disturbed flows equations which here are the main three:

- the statistical method, designated under symbol RANS-"Reynolds Averaged Navier-Stokes", uses Navier-Stokes equations by passing at the average, the result is therefore a loss of information;

- Large Eddy Simulation (LES) - this simulation's major idea is to calculate the contributions to the large scales of flow while modelling is reserved for structures whose size is less than a characteristic dimension calculation mesh.

- Direct Numerical Simulation (DNS) - exclude any modelling of turbulent agitation. We operate numerically at the resolution of Navier-Stokes equations. Thus, obtaining statistical data on the flow is reported after the resolution. This method requires more memory space and calculation time than large scales simulation method and is therefore more expensive. It still belongs to the domain of research and is useful to analyze phenomena associated with small scales and in particular sub-meshes schemas. It can be said that this method allows some real "numerical experiences" whose results can both give complete knowledge drawn from traditional physical experiments or provide support for developing models for other categories.

The phenomenon of turbulence, although chaotic, does not occur outside the laws and the known equations for fluids flow. The general framework where we model fluid flow is given by a set of equations: the continuity equation, equation of quantity of movement, equation of energy and perfect gas law. The unknown of the system are: velocity, with its three components u, v and w, the pressure p, the fluid density ρ and temperature T. All these parameters are functions variable in time and space [4].

The continuity equation is expressed by the relation:

$$\frac{\partial \rho}{\partial t}+\frac{\partial\ \rho u}{\partial x}+\frac{\partial\ \rho v}{\partial y}+\frac{\partial\ \rho w}{\partial z}=0, \tag{1}$$

where ρ is the fluid density and u, v and w are the components of the fluid velocity.

The continuity equation may be written in the vector form:

$$\frac{\partial \rho}{\partial t}+\rho \mathrm{div} V+V \mathrm{grad} \rho=0. \tag{2}$$

In the case of a constant fluid volume, density is independent of time and space variables. Equations of quantity of movement are the equations of the balance between forces of inertia, mass forces, pressures and constraints due to friction. The vector form of the quantity of movement equation is expressed by:

$$\rho \frac{dV}{dt}=\rho f-\mathrm{grad} p+\frac{1}{3}\mu \mathrm{grad}\ \mathrm{div} V\ \ +\mu \Delta V, \tag{3}$$

where f is the external mass force acting on the unit of mass and μ is the dynamic viscosity.

Momentum equations can be projected in the Cartesian coordinates for an incompressible fluid neglecting the external mass forces:

$$\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}=-\frac{1}{\rho}\frac{\partial p}{\partial x}+\frac{\mu}{\rho}\ \frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\ , \tag{4}$$

$$\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}=-\frac{1}{\rho}\frac{\partial p}{\partial y}+\frac{\mu}{\rho}\ \frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\ , \tag{5}$$

$$\frac{\partial w}{\partial t}+u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}=-\frac{1}{\rho}\frac{\partial p}{\partial z}+\frac{\mu}{\rho}\ \frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\ , \tag{6}$$

where μ is the dynamic viscosity.

Large Eddy Simulation aim is to solve the complete Navier-Stokes equations while reducing the number of node of the mesh necessary to resolve a fully turbulent flow. Reynolds numbers encountered in natural turbulent flow prohibit current computers perform a numerical simulation of all scales involved. The idea of LES is to practice a cut to a wavelength $k=k_c$ (figure 1). We simulate directly large scales corresponding to $k<k_c$ and we model small scales corresponding to $k > k_c$ [1].

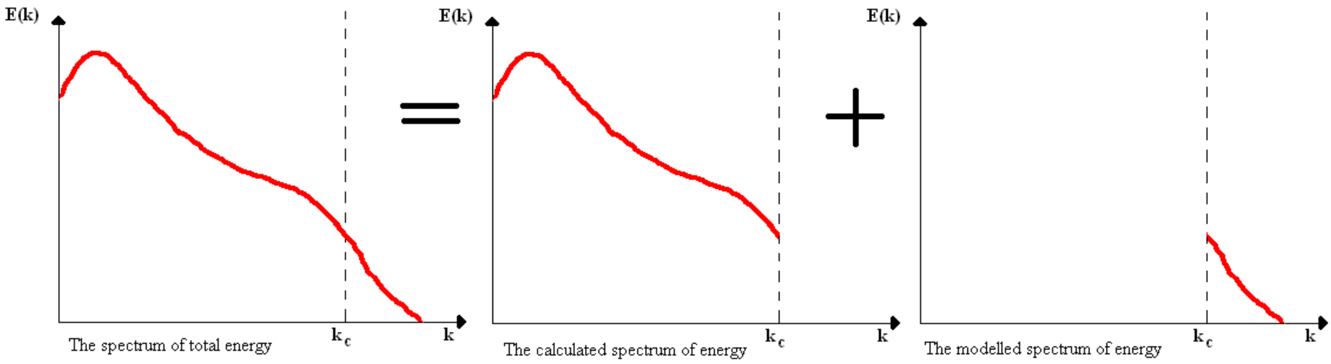

Figure 1. Decomposition of the energy spectrum of the solution associated to the LES simulation

Large scales (see figure 3) are less universal and are more dependent on the geometry of the flow. Small scales are less affected by the geometry of the flow, so it should be easier to find universal models to represent their effects. Velocity field is in this case divided into the sum of a velocity field corresponding to the large scales and a sub-mesh velocity field that represents the wavelengths not picked up by the mesh. It should be noted that this separation between different scales is not related to an average operation statistics [7].

The formalization in mathematical terms of this separation of scales is to express the passage through a low pass filter in frequency and in number of wave. The application of this filter to the Navier-Stokes equations allows obtaining the constituent mathematical model for the simulation of the large scales. Part of the resulting terms can be calculated directly from resolved scales, the other must be modelled.

The separation of scales is performed by the application of a high pass filter in spatial scale and low pass in frequency to the exact solution. Thus, for any quantity associated $\varphi(x,t)$ with the turbulent movement it sets the quantity resolved $\varphi(x,t)$ by:

$$\varphi(x,t) = \int_V G_\Delta(x-x')\,\varphi(x',t)\,dx', \tag{7}$$

where the convolution kernel $G_\Delta(x-x')$ is characteristic to the used filter that is associated with cut-off in space scales Δ (bandwidth filter) [1].

This paragraph is realised to bypass the constituent equations of simulation techniques of the large scales, i.e. filtered Navier-Stokes equations. Navier-Stokes system variables are written as follows:

$$u_i(x,y,z,t) = u_i(x,y,z,t) + u'_i(x,y,z,t), \tag{8}$$

$$p(x,y,z,t) = p(x,y,z,t) + p'(x,y,z,t). \tag{9}$$

This modelling must not only have a role of dissipation of the surplus energy not absorbed by the small structures which are not resolved by the code, but also take account of the energy return - that exists in some parts of a turbulent flow of these small structures - to the resolved scales of the field.

The system of Navier Stokes equations for the non-filtered field is:

$$\frac{\partial u_j}{\partial x_j} = 0, \tag{10}$$

$$\frac{\partial u_i}{\partial t} + u_j\frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial p}{\partial x_i} + 2\frac{\mu}{\rho}\frac{\partial S_{ij}}{\partial x_j}. \tag{11}$$

Assuming that the filter characteristics allow the use of commutative property between operator filtering and operator derivative, after we apply the filter to the equations of Navier – Stokes, we obtain the following equations [1]:

$$\frac{\partial u_j}{\partial x_j} = 0, \tag{10}$$

$$\frac{\partial u_i}{\partial t} + \frac{\partial u_i u_j}{\partial x_j} = -\frac{1}{\rho}\frac{\partial p}{\partial x_i} + \frac{\partial\left(2\frac{\mu}{\rho}S_{ij} - \tau_{ij}^{sm}\right)}{\partial x_j}, \tag{11}$$

where: $\tau_{ij}^{sm} = \overline{u_i u_j} - u_i u_j$ is the constraints tensor of the sub-mesh;

$S_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)$ is the tensor of the filtered deformations rate.

We can write the constraints tensor of sub-meshes if we introduce decomposition in the first term:

$$\tau_{ij}^{sm} = \overline{u_i u_j} - u_i u_j = L_{ij} + C_{ij} + R_{ij}, \tag{12}$$

where

$L_{ij} = \overline{u_i u_j} - u_i u_j$, the Leonard term;

$C_{ij} = \overline{u_i u'_j} - \overline{u'_i u_j}$, the term of the crossed constraints;

$R_{ij} = \overline{u'_i u'_j}$, the term of Reynolds constraints.

The non-linear term is now fully deconstructed depending on the amount u_i and the fluctuation u'_i.

Large scales simulation is a technique for reducing the number of degrees of freedom for the solution. This reduction operates separate scales that conduct to obtaining the exact solution into two categories: large scales and sub-meshes scales.

Decreasing of the solution complexity is obtained by retaining only the large scales for the numerical resolution. Information related to small scales is obtained after an implementation of a specific model. Subsequently, all terms that make them appear, i.e. the term in u' in physical space, may not be calculated directly. These terms are grouped together in the sub-mesh tensor. However, to ensure that the dynamics of the scales resolved remains correct, the term of sub-mesh must be taken into account.

The sub-mesh scale modelling consists in expression of this term function of the resolved scales. In this case we have to use additional hypothesis, derived from knowledge on mechanics of the fluid that permit to link the existence of sub-mesh modes to certain properties of resolved scales.

3. Simulation of aircraft wake vortex

During the evolution of a pair of contra-rotating longitudinal vortex in the presence of an external turbulence, we observe the generation of the instability at large wavelength, named Crow instability. We have to study the behaviour of a pair of vortices in the presence of such disturbance.

In the aim to achieve this disturbance, we disturb the vortices centre position. As parameters to this disturbance we use: the magnitude of the disturbance Amp, the angle of the disturbance plane θ, the phase difference ϕ and the longitudinal wave number k (figure 2). In the transverse plane, disturbed vortex centre coordinates are [7]:

$$y_{c_{\text{pert}}} = y_{c_{\text{nepert}}} + Amp \sin\ kx + \phi\ \ \cos\theta, \tag{13}$$

$$z_{c_{\text{pert}}} = z_{c_{\text{nepert}}} + Amp \cos\ kx + \phi\ . \tag{14}$$

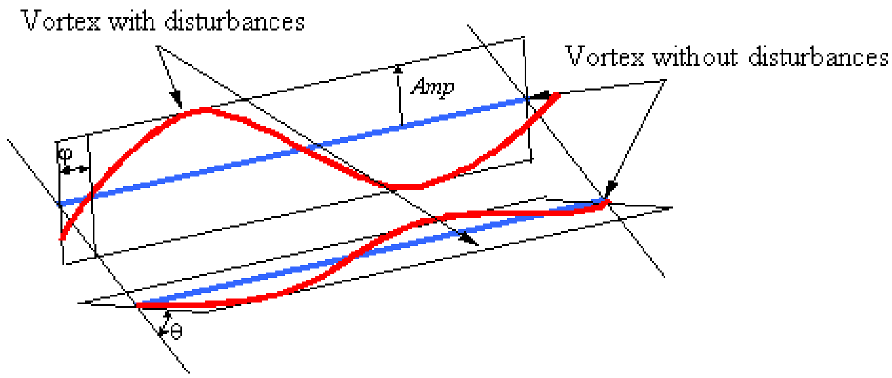

Figure 2. Crow type disturbance

To initiate the calculations, we use a vortex dipole model, whereas two longitudinal vortices with circulations $\Gamma_{01} = -\Gamma_0$, $\Gamma_{02} = \Gamma_0$, $\Gamma_0 = 1$with the same radius $r_0 = 0.2 \cdot b$, located at a distance $b = 1$ to each other, b being the distance between the edges of the wings. Basis positions for vortices cores are given by the coordinates $y_{c_1}, z_{c_1} = 2.5, 2.5 \cdot b$ and $y_{c_2}, z_{c_2} = 3.5, 2.5 \cdot b$, in the transversal planeOyz. This position is disturbed by the explicitly introduction of the Crow instability. The Crow disturbance parameters used are the maximum amplitude $Amp = 0.2$, the longitudinal wavelength $\lambda = 7.5$ and the phase difference $\phi = 0$. Disturbance plan is rotated as 45 degrees.

The computation field, with a regular mesh, is a rectangular box with dimensions: $L_x = 7.5 \cdot b$ with $n_x = 95$ points, $L_y = 6 \cdot b$ with $n_y = 95$ points, $L_z = 5 \cdot b$ with $n_z = 64$ points (577,600 calculation meshes). The centre of the vortex is represented by 7 points in the direction y and by 6 points in the z direction. The used boundary conditions are periodic in x (the longitudinal axis of the vortex) and z directions (the vortex pair descent direction) and symmetry in y. The characteristic time for the evolution of a pair of the vortex is based on the movement of descent of the two vortices,

$$t_0 = \frac{2\pi b_0^2}{\Gamma_0} = 6.28 \text{ s}. \tag{14}$$

The flow parameters evolution is expressed using the dimensionless time,

$$T = \frac{t}{t_0}. \tag{15}$$

Figures 3 and 4 present the iso-surfaces of vorticity function of the characteristic time. A pair of longitudinal vortex, influenced explicitly by a Crow disturbance, will generate the vortex rings. We observe that in absence of other external disturbances, vortex rings deform and after 20 seconds ($T = 3.2$), the computation field will not be large enough to process correctly the evolution of these vortex rings.

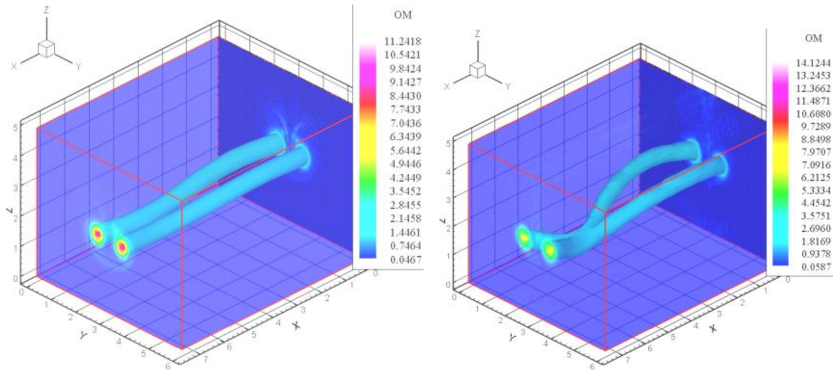

Figure 3. Iso-surface of vorticity ω after T= 0.31 and T= 1.27

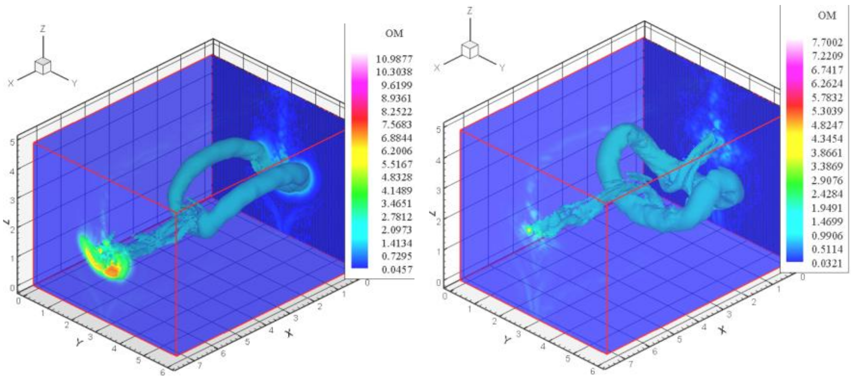

Figure 4. Iso-surface of vorticity ω after T= 1.90 and T= 2.54

4. Conclusions

This paper contains studies regarding the mathematical modelling of turbulent aerodynamic phenomena. As physical phenomena that take place in aircraft wakes can be assimilated to turbulent flows, it was necessary to realize a study of the main theoretical methods and models that can be used in numerical simulation of turbulent phenomena. The choice of the most appropriate method is based on the analysis of the turbulent energy spectre. These methods are based on finding a solution through the method of finite volume of fundamental equations of turbulent fluid flow, the Navier-Stokes equations.

The Large Eddy Simulation (LES) method of modelling turbulent flow consists of directly simulating large turbulent structures, the small ones being modelled by specific methods. The reasoning of this method is based on the fact that large turbulent structures are directly influenced by the geometrical characteristics of the studied situation, while small structures have a universal character, and the errors introduced by modelling them are insignificant. The advantage of this method is that it offers results as precise as those obtained through the direct numerical simulation method, but using a smaller amount of calculus numbers, therefore a smaller calculus power. The study shows the receptivity of a pair of longitudinal vortices contrarotatifs against the Crow disturbance. Through the perturbation of vortices filaments with a Crow disturbance, unsteadiness is inducted with big wavelength, that leads to the destruction of linear vortices and the formation of vortices rings

References:

1. Calmet I., Magnaudet J., Large eddy simulation of high-Schmidt number mass transfer in a turbulent channel flow, Physics of Fluids Review, Volume 9, Issue 2, pag. 438-455, American Institute of Physics, Maryland, USA, 1996;
2. Giovannini A., Boisson H., Moldoveanu C.E., Effect of wind conditions on the evolution of an aircraft vortex wake model near the ground, Proceedings of the 6th International Conference on Computational Fluid Dynamics, pag. 94, St. Petersburg, Rusia, 12-16 iulie 2010;
3. Moldoveanu C.E., Giovannini A., Boisson H., Turbulence receptivity of longitudinal vortex-dominated flows, Lecture Notes in Computational Science and Engineering, Vol. 69: BAIL 2008 - Boundary and Interior Layers, Proceedings of the International Conference on Boundary and

Interior Layers- Computational and Asymptotic Methods, Springer Berlin Heidelberg, pag. 227 – 236, ISBN: 978-3-642-00604-3, 2009;
4. Moldoveanu C.E., Giovannini A., Boisson H., Moraru F., Large Eddy Simulation of vortex pair submitted to ground effect, New Aspects of Engineering Mechanics, Structures, Engineering Geology, Proceedings of the WSEAS International Conference on ENGINEERING MECHANICS, STRUCTURES, ENGINEERING GEOLOGY (EMESEG '08), Heraklion, Crete Island, Greece, 22-24 iulie 2008, pag. 265-270, ISSN 1790-2769, ISBN 978-960-6766-88-6;
5. Moldoveanu C.E., Safta D., Şomoiag P. – Analyze of the Counter-Rotating Vortices Pair Stability under the External Turbulence Influence, Lecture Notes in Engineering and Computer Science, WCE2008 Proceedings of The World Congress on Engineering 2008, The 2008 International Conference of Mechanical Engineering, Londra, United Kingdom, 2-4 iulie 2008, Volume II, pag. 1463-1467, ISBN 978-988-17012-3-7;
6. Moldoveanu C., Boisson H., Giovannini A., Moraru F.“Numerical simulation of longitudinal counter-rotating vortices pair in interaction with ground”, Fluid Mechanics and Aerodynamics, Proceedings of the 5th IASME/WSEAS International Conference on Fluid Mechanics and Aerodynamics (FMA’07), Vouliagmeni, Grecia, 25-27 august 2007, pag. 223-228, ISSN 1109-2769, ISBN 978-960-8457-99-7 ;
7. Moldoveanu C.E., Boisson H., Giovannini A., Receptivity of a longitudinal contrarotating vortices in an external flow: a numerical experimentation, Conference on Boundary and Interior Layers, BAIL 2004, Toulouse, Franța, 5-9 iulie 2004.