MATHEMATICAL MODELS USED TO STUDY
THE AIRCRAFT WAKE VORTICES

Cristian-Emil MOLDOVEANU*, Pamfil ŞOMOIAG*,
Pierre FERRIER**, Héloïse DAUDE**

*Military Technical Academy, Bucharest Romania, ** University Paul Sabatier, Toulouse, France

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 take-off and landing. This paper presents the mathematical models used to simulate the aircraft wake vortex behavior.

Keywords: aircraft, wake vortex, turbulence

1. INTRODUCTION

Research conducted on the wake turbulence of aircrafts is determined especially by a series of economic, ecological and flight safety and security demands. These vortices are responsible for resistance to advancement induced by the lift force, which represents approximately a third of the total resistance to advancement of the aircraft. Considering the actual importance of air transport, an induced decrease with only a small percentage in resistance to advancement allows an annual significant cutback in fuel consumption (a few billions of liters of fuel).

Induced resistance is an important characteristic that is taken into consideration when projecting a new wing. Therefore, the weaker the intensity of the wake turbulence, the weaker the resistance induced by the lift force, thus decreasing fuel consumption [1]. This raises the problem of decreasing the intensity and coherence of the wake turbulence of aircrafts, in order to decrease the resistance induced by the lift force [2]. In this case it is recommended to reduce the distribution of the lift force near the end of the wing, by using longer wings and decreasing the incidence or the surface towards the end of the wing.

Wake turbulence of aircrafts also has a negative ecological consequence, because of its role in the dispersion of polluting particles in the atmosphere and through the contribution of the artificial cloud created by condensation, to the energetic balance of the planet. The persistence of the artificial cloud created in the air is directly connected to the duration of vortices inside wakes.

The most important consequence of wake turbulence consists of the risk of air incidents due to the encounter between an aircraft and the wake generated by a preceding aircraft [3].

This situation is more frequent in the take-off and landing phases, when aircrafts are encountered, during movement, one behind the other, at a small enough distance and when
interaction phenomena between the wake and
the ground can lead to an increase of the
intensity of wake vortices and, thus, to a
probability of air accident production with
devastating consequences.

2. RISK ASSESSMENTS OF ACCIDENTS
PRODUCED AS A RESULT OF THE
AIRCRAFT WAKE VORTEX

The main effect of wake turbulence on an
aircraft is the induced rolling motion that can
even lead to an overturn. These phenomena
can become extremely dangerous during the
take-off or landing phases when the aircraft
moves at a low altitude and can’t recover.
Aircrafts that have small wing spans are the
most affected [4]. The effect of the wake
depends on various factors, of which we
remind the weight and the span of the aircraft
that enters in the action range of the wake and
the relative position between the aircraft and
the wake turbulence. Due to the action of the
wake turbulence, numerous air accidents
happened along time:

- 30 May 1972 - The McDonnell Douglas
DC-9-14 Aircraft crashed on the Southwest
International Airport in Fort Worth, Texas,
being affected by the wake turbulence
generated by a McDonnell Douglas DC-10
aircraft. Following this incident, a series of
regulations regarding the minimal separation
distances between aircrafts have been
introduced;

- 20 September 1999 - a JAS39 Gripen
aircraft passed through the wake turbulence of
another aircraft of larger proportions during a
military maneuver, crashing in a lake;

- 12 November 2001 - an Airbus A300
aircraft crashed on New York airport, under
the action of the wake turbulence generated by
a Boeing 747 aircraft which had taken off two
minutes before;

- 4 November 2008 - LearJet 45 XC-VMC
aircraft crashed near the international airport in
Mexic. Subsequently, it was proven that this
incident was due to the action of the wake
turbulence of a Boeing 767 aircraft;

- 25 February 2009 - the Boeing 737-800
aircraft of the Turkish Airlines company
crashed near the Schiphol airport in
Amsterdam, being affected by the wake
turbulence of a Boeing 757 aircraft, that had
landed on the same landing run two minutes
before;

- 2 March 2009 - according to press
information, the Control Tower of the Henri
Coanda airport in Otopeni discovered in due
time that the landing trajectory of the ATR
aircraft of the TAROM company intersected
that of a Boeing 727 aircraft of the same
company. At the last moment, the employees
of the control tower have cancelled the landing
procedure, and the plane was forced to
postpone landing, in order to avoid passing
through the action range of the wake
turbulence of the first aircraft.

In order to avoid air incidents caused by
the action of wake turbulence a series of
regulations have been established to define the
minimal time intervals between two
landings/take-offs of two aircrafts, depending
on their proportions, so that the wake of the
first aircraft doesn’t affect the second aircraft
significantly (fig.1). For example, no other
aircraft should take-off or land behind a
Boeing 747, no sooner than two minutes,
which corresponds to a distance of
approximately 7.2 km between the aircrafts.
This distance becomes greater if the following
aircraft is from a smaller category [5].

Fig. 1. Distance of separation between two
aircrafts

The minimal time intervals established by
the air authorities led to a limitation in air
traffic. Therefore, modern airports are dealing
with a highly important problem that is
optimizing air traffic so that it allows the
circulation of a larger number of aircrafts on the take-off/landing run, but without endangering them [5].

Research on wake dynamics have been the object of important programs financed both by the European Union and by other international organizations.

3. MODELS FOR WAKE VORTEX

3.1 Rankine vortex. A first approach to longitudinal vortex is the two-dimensional isolated vortices type Rankine. This vortex is a classic model characterized by a rotation in the vortex core. It corresponds to a constant vortices ($\omega$) distribution in a cylindrical tube infinite and with a radius $r_0$. The vortices are equal to zero outside of the tube (fig. 2).

The vorticity for the Rankine vortex will have the distribution:

$$\omega(r) = \begin{cases} \frac{\Gamma_0}{\pi r_0^2}, & 0 < r \leq r_0 \\ 0, & r > r_0 \end{cases}$$

The tangential speed for the Rankine vortex will have the relation:

$$u_\theta(r) = \begin{cases} \frac{1}{r}, & 0 < r \leq r_0 \\ \frac{1}{r_0} \int_0^r \omega_0 r' dr', & r_0 < r \end{cases}$$

(2)

$$u_\theta(r) = \frac{1}{2r} \int_0^r \omega_0 r'^2 dr', 0 < r < r_0$$

(3)

The Rankine vortex model which is presented here consists of a solid rotation of the heart of the vortex and a potential flow. In this model the viscosity isn’t taken into account and also this model is not an exact solution of Navier-Stokes equations [6].

3.2 Lamb-Oseen vortex. In analyzing the influence of the viscosity on the vortex dynamics we used another longitudinal vortex, Lamb-Oseen vortex model. This situation allows us to obtain an analytic solution. The flow of a single vortex isolated as Lamb-Oseen admits a revolution symmetry. We chose also cylindrical coordinates. We used Navier - Stokes equations to solve this problem [7].

The vorticity for a vortex will have this expression:

$$\omega(r, T) = \frac{\Gamma_0}{\pi r_0^2} \left( 1 + \frac{1}{1 + T r^2} - \frac{1}{1 + T r_0^2} \right)$$

(4)

where we use the reduced time $T$, and we note the viscous time with $t_v$:

$$T = \frac{t}{t_v}, t_v = \frac{r_0^5}{4 \nu}$$

(5)

Also, the tangential velocity (fig. 3) of a vortex will have this expression:
The Lamb-Oseen vortex is a simple model of longitudinal vortex frequently used to model the wake of the aircraft. It provides a good numerical representation of longitudinal vortices.

3.3 Lamb-Chaplygin model for wake-vortex pair. Preceding models are used for solitary vortices. The non-linearity of the Navier-Stokes equations imposes some problems in the case of the superposition of elementary isolated vortices. The model of Lamb-Chaplygin takes into account the both the high wake vortices. Besides, in this model, it is not held counts of the viscosity, what removes considerably results of the reality.

Lamb-Chaplygin model consists of a counter-rotating wake-vortex pair with the vorticity concentrated in the circle with the radius $R$ (fig. 4):

$$\omega(r, \theta) = \begin{cases} \frac{-\mu_1^2}{R^2} \psi(r, \theta), & r \leq R \\ 0, & r > R \end{cases}$$

where:

$U$ – velocity of vortex-pair propagation;

$J_0, J_1$ – Bessel functions;

$\mu_1 = 3.8317$, the first solution of $J_1$;

$$\psi(r, \theta) = \begin{cases} -\frac{2U \cdot R}{\mu_1} J_1(\mu_1 \cdot \frac{r}{R}) \sin \theta, & r \leq R \\ -U \cdot r \left(1 - \frac{R^2}{r^2}\right) \sin \theta, & r > R \end{cases}$$

The solution obtained by using of superposition of two simple vortices is not a solution of the equations of Navier-Stokes due to the non-linearity of these equations.
After superposition of two vortices we should use a calculation of adaptation to remove this defect \[10\]. Fig. 6 and 7 present the velocity and pressure distribution obtained using the superposition methods of two counter-rotating Lamb-Oseen vortices.

\[
\begin{align*}
 v(y,z) &= -\frac{\Gamma_0}{2\pi} \frac{z-z_c}{r^2} \left[ 1 - e^{-i(r/r_c)^2} \right], \\
 w(y,z) &= \frac{\Gamma_0}{2\pi} \frac{y-y_c}{r^2} \left[ 1 - e^{-i(r/r_c)^2} \right],
\end{align*}
\]

where \( r = \sqrt{(y-y_c)^2 + (z-z_c)^2} \).

Fig. 7. Pressure distribution in the case of counter-rotating vortex pair

Solutions are defined within calculation, also to have solutions that correspond to the reality, this implements a mathematical model that will interact with the solution proposed for the closer to reality. After a sufficient number of iterations of the simulation of adaptation, speeds and pressures field obtained can be considered as the Navier Stokes equations solution \[11\].

3.5 Counter-rotating vortices using rows of vortices model. The method of modeling a pair of counter-rotating vortices by the superposition method describes the flow in the infinite medium. The numerical simulations are often done in a finite computation field. In this case the rows of vortices are the most appropriated method \[9\].

To create the rows of vortices are used Lamb-Oseen vortices (fig. 8). For a single vortex with the coordinates of its heart in the plane \( Oyz, (y_c, z_c) \), the velocity field has the following distribution:

\[
\begin{align*}
 v(y,z) &= -\frac{\Gamma_0}{2\pi} \frac{z-z_c}{r^2} \left[ 1 - e^{-i(r/r_c)^2} \right], \\
 w(y,z) &= \frac{\Gamma_0}{2\pi} \frac{y-y_c}{r^2} \left[ 1 - e^{-i(r/r_c)^2} \right],
\end{align*}
\]

where \( r = \sqrt{(y-y_c)^2 + (z-z_c)^2} \).

Fig. 8. Rows of vortices

We consider a vertical row of vortices at a distance. To determine the velocity distribution for this row using the stream function as follows:

\[
f(\xi) = -\frac{\Gamma_0}{2\pi} \ln(\xi - \xi_c), \tag{10}\]

where \( \xi = y + zi \) et \( \xi_c = y_c + z_c i \).

After computing for \( n \to \infty \) vortices we get the rows of vortices:

\[
\begin{align*}
 v(y,z) &= -\frac{\Gamma_0}{2\pi} \frac{\sin[2\pi(z-z_c)/b]}{2b \cosh[2\pi(y-y_c)/b] - \cos[2\pi(z-z_c)/b]}, \\
 w(y,z) &= \frac{\Gamma_0}{2\pi} \frac{\sinh[2\pi(y-y_c)/b]}{2b \cosh[2\pi(y-y_c)/b] - \cos[2\pi(z-z_c)/b]},
\end{align*}
\]

In the middle row we must remove the solution corresponding computational domain:

\[
\begin{align*}
 v(y,z) &= -\frac{\Gamma_0}{2\pi} \frac{z-z_c}{r^2} \left[ 1 - e^{-i(r/r_c)^2} \right], \\
 w(y,z) &= \frac{\Gamma_0}{2\pi} \frac{y-y_c}{r^2} \left[ 1 - e^{-i(r/r_c)^2} \right].
\end{align*}
\]

It has been added \( m \) shifted images in the horizontal direction. The final solution is:
\[ S(y_c,z_c) = S_o(y_c,z_c) + S_r(y_c,z_c) - \sum_m S_p(y_c \pm a \cdot m, z_c) \]  

(13)

where:
- S – final solution;
- \( S_0 \) – solution for the central vortex pair (9);
- \( S_r \) – solution corresponding to the row of vortex (11);
- \( S_p \) – solution corresponding to computational field (12).

5. CONCLUSIONS & ACKNOWLEDGMENT

The presented models can be used as initial condition for the numerical simulations of the behavior of the isolated wake vortex or of the wake vortex counter-rotating pair [8]. The numerical simulation can be realized using the Direct Numerical Simulation or Large Eddy Simulation methods [12].

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 Direct Numerical Simulation (DNS) method of modeling turbulent flow has the advantage of being very precise, but it’s necessary to realize simulations on a very large number of calculus points, which obviously involves using a high calculus power.

The Large Eddy Simulation (LES) method of modeling turbulent flow consists of directly simulating large turbulent structures, the small ones being modeled 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 an 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.

REFERENCES