Experimental simulation of the flow around the keel of a yacht

Renata M. B. Chaves*

Mariana F. O. F. Lima**

Alexandre Teixeira P. Alho**
**Departamento de Engenharia Naval e Oceanica
Universidade Federal do Rio de Janeiro

Atila P. Silva Freire*
*Programa de Engenharia Mecânica, COPPE/UFRJ
C.P. 68503, 21945-970, Rio de Janeiro, Brasil

Abstract. *The hot-wire anemometry is used to characterize the flow around the keel of a yacht. The experiments were conducted in a low speed wind tunnel, in four downstream stations. Through the hot-wire anemometry, the following properties of the flow were assessed: mean velocity, longitudinal velocity fluctuations and the moments of second, third and forth order. The purpose here is to provide data for an adequate modeling of the process. In a companion paper, the applicability of different turbulent models for the adequate description of the flowaround the keel of a yacht will be discussed. The following models will be considered considered: an eddy viscosity model, and two Reynolds stress model. All data are compared with the experiments performed through the hot-wire anemometry.*

Keywords: *Ship technology, keel, hot-wire anemometry.*

1. Introduction

In a host of practical applications, the flow of a fluid is set to develop around three-dimensional obstacles composed of many parts. The examples are many in external aerodynamics, turbomachinery, flows over buildings, electronic component cooling, in animal flight and swimming, just to mention a few applications.

In all cases, the intrinsic geometry of the problem under interest gives rise to very complex flow configurations. In fact, when many parts of an equipment, or a machine, are assembled together, several wakes will result that will interact with each other in a very complex way. Regions where parts of a machine are attached to a surface also produce complex flow situations due to three-dimensional separations.

For high performance vehicles, unfortunately, all the above remarks remain valid. Even very highly streamlined vehicle have to resort to appendices to perform the most various tasks. These appendices will, then, in most cases provoke flow interferences resulting from wakes or separations.

In this work, our main concern lays with ship technology applications. In particular, we will investigate the flow around the keel of a yacht. Using the hot-wire anemometry, the flow pattern downstream of a keel in a wind tunnel will be assessed. Measurements will be presented for the mean velocity and the turbulent moments of second, third and fourth order for one downstream measuring station. In fact, laboratory measurements were made for four different stations, however, for the sake of brevity, the results of just one equation will be presented here. The hot-wire anemometer is a low cost equipment with a very high frequency response, relative small size, low noise levels, high accuracy and a continuous analog signal. For this reason, there is no surprise such instrument is judged ideal for measurements in low and moderate turbulence intensity flows.

A companion paper will present some numerical simulations of the flow around the keel. Turbulent models based on the eddy viscosity concept will be investigated as well as models that rely on direct modeling of the Reynolds stress (RSM).

When a flow approaches a region of adverse pressure gradient, the boundary layer is prone to separate and generate multiple horseshoe vortices. In flows near a junction, a stagnation point occurs on the surface that separates the downstream/upstream regions and flows that pass around the sides of the obstacle that define the function. The amount of flow that is displaced sideways, of course depends on several factors including the state of the boundary layer through its momentum thickness, the flow Reynolds number, the free-stream level, the roughness of the surface, the vortices that are generated. Of course, a faithful simulation of the problem would have to embed in its model all these variables. This work will provide data for mean velocity and turbulent quantities so that future numerical simulations of the flow around a keel can be rightly assessed.

2. Further remarks on junction flows

Next we will present a short review on flow near a junction point.

The subject of junction flows received a thorough review by Simpson (2001). The author discussed the physics of flow separation at junctions for both laminar and turbulent flows. The three-dimensional separation with resulting horseshoe vortices that are generated at junctions were explained in phenomenological terms. Calculation flow methods were also reviewed. For that matter, Simpson advices one to use methods that can capture large-scale chaotic vertical motions. Works about the control, modification, or elimination of horseshoe vortices were also reviewed.

The control of horseshoe vortices was extensively examined by Barberis et al. (1998). Basically two methods were studied: i) the modification of pressure field induced by the obstacle by changing its geometry with a strake, and ii)the implementation of suction through a hole located near the separation line. The study was experimental, and was conducted in a subsonic wind tunnel. The surface flow properties were characterized by using a viscous coating to allow a visualization of the skin friction line patterns. Quantitative results were obtained with Laser Doppler Velocimetry (LDV). The paper showed that the geometrical extent of the separation region is reduced and the separation line is much close to the leading edge when a strake is fitted to the model. Using active control, that is, producing suction through a hole located in the plane of symmetry in front of the leading edge of the obstacle, the vortex size is reduced and its location displaced towards the origin of the obstacle.

Cummings et al. (2003) considered the computational challenges involved in the numerical computation of flows under a high angle of attack. When vertical and massively separated flows occur, appropriate numerical algorithms have to be used. In addition, grids must be created that provide accurate simulations of the flow field. The paper proposes solutions based on hybrid turbulence models.

Non-linear eddy viscosity and algebraic stress models can provide a link between linear eddy viscosity models and the full differential Reynolds stress model. Gatski and Jongen (2000) reviewed several models providing a perspective view on their various similarities and differences. Their link with Reynolds stress models and the best predictive capability over standard linear eddy viscosity models was also addressed.

3. Experiments

3.1 The hot-wire anemometer

Hot-wire anemometry (HWA) is based on the convective heat transfer process that takes place when a heated wire is exposed to a fluid flow. Because typical sensors are less than 5 μm in diameter and are made to withstand high temperatures. Any change in fluid flow condition that affects the transfer of heat from the wire to the medium will be sensed immediately by a constant HWA system. HWA can then be used to measure the velocity and temperature of the flow, concentration and phase discrimination.

In general, one of the most important aspects of thermal anemometry is the accurate interpretation of the anemometer signal. The main purpose of any sensor calibration is to determine, as accurately as possible, the relationship between the anemometer output voltage and the physical property under consideration, in this case velocity or temperature. However, the direct output from all practical calibration procedures is raw calibration data, which will contain measurements uncertainties. An additional complication is that the true calibration curve of the probe is not known, and furthermore it depends on particular characteristics of each experiment.

Among the several possible methods that can be devised to characterize the velocity and temperature dependence of thermal anemometers signals, the linear correction method is the simplest. Through this procedure,

the heat transfer from the probe is assumed to be proportional to a product of the temperature difference T_w- T_a and a function of the velocity, where T_w is the temperature of the heated wire and T_a is the ambient temperature. The output voltage, *E*, of a constant temperature hot-wire anemometer can hence be represented by:

$$\frac{E^2}{T_w - T_a} = A + BU^n$$

The most accurate way of establishing the velocity and temperature sensitivity of a constant temperature hot-wire probe, operated at a fixed hot resistance, R_w, is to measure the anemometer output voltage, *E*, as a function of the velocity, *U*, and fluid temperature, T_a. This type of calibration is often carried out by performing a velocity calibration at a number of different fluid velocities and temperatures.

3.2 Experimental set up

The flow around the keel of a yacht was studied in the low-turbulence wind tunnel of the Laboratory of Turbulence Mechanics of PEM/COPPE/UFRJ. This tunnel presents turbulence intensity levels of the order of 0.2% and can be set to run at velocities that can reach 13 m/s. The test section is 4 m long and the cross section area is 0.30 x 0.30 m. The tunnel has honeycombs and screens to control the turbulence levels so as to guarantee a uniform flow. The computer-controlled traverse gear is two-dimensional and capable to position sensors with an accuracy of 0.02 mm

The experiments were conducted in a controlled environment, with the laboratory temperature set to 18.0 ^oC +/- 0.5 ^oC.

The general features of the wind tunnel are shown below.

- Circuit: open.
- Test section: 0.30 m high, 0.30 m wide and 4 m long.
- Wind speed: continuously variable from 0.5 to 16 m/s.
- Longitudinal pressure gradient: adjustable to zero by means of an adjustable ceiling.
- Turbulence intensity: below 0.2%.
- Incoming flow temperature: variable from 20 to 35 ^{o}C.

A general view of the low-turbulence wind tunnel is shown in Figure 1. Details of the keel assembly are shown in Figure 2

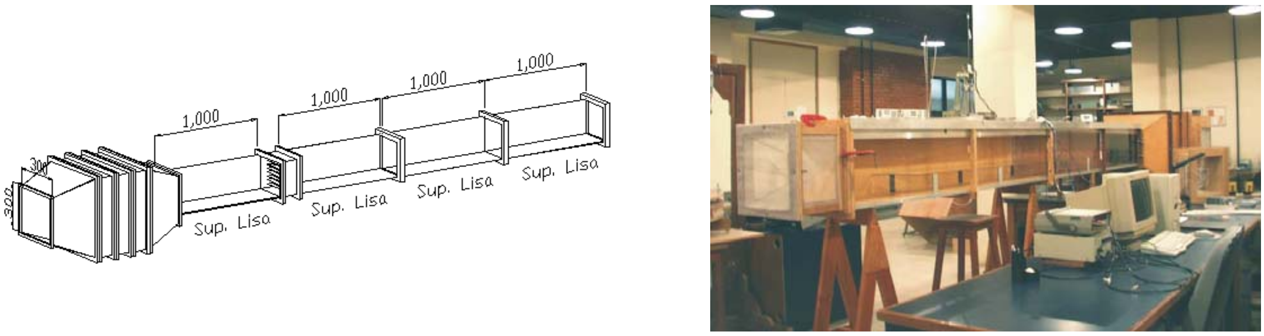

Figure 1. General view of wind tunnel. Dimensions in millimeters.

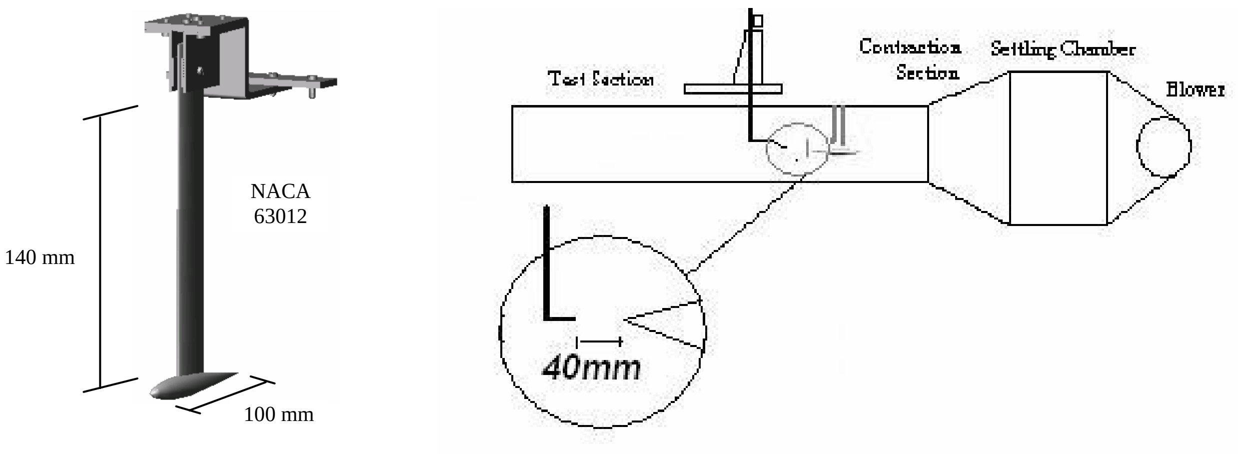

Figure 2. Assembly details of the keel. Model keel is seen on the left. Keel in the wind tunnel is seen to the right.

For the present measurements, a DANTEC 55M01 main unit together with a 55M20 constant current bridge was used. The boundary layer probe was of the type 55P11. A Pitot tube, an electronic manometer, and a computer controlled traverse gear were also used. In getting the data, 10.000 samples were considered. An uncertainty analysis of the data was performed according to the procedure described in Kline (1985). Typically the uncertainty associated with the velocity and temperature measurements were: U = 0.0391 m/s precision, 0 bias (P=0.95).

To obtain accurate measurements, the mean and fluctuating components of the analogical signal given by the anemometer were treated separately. Two output channels of the anemometer were used. The mean velocity profiles were calculated directly from the untreated signal of channel one. The signal given by channel two was 1 Hz high-pass filtered leaving, therefore, only the fluctuating velocity. The latter signal was then amplified with a gain controlled between 1 and 500 and shifted by an offset so as to adjust the amplitude of the signal to the range of the A/N converter.

4. Results

The three-dimensional mean velocity fields for the two investigated stations are shown in Fig. 3 together with a contour line plot.. The turbulent fields are shown in Fig. 4.

The figures show a deflection of the region of higher turbulence action to the left side of the model. This clearly, results from existing asymmetries in the constructed model.

The noteworthy point is the large suppression in mean velocity in the turbulent region resulting from the trailing vortices generated at the junction point defined by the keel and its support. In fact, the mean velocity is seen to decrease from its undisturbed value of 6 m/s to about 3.8 m/s. Of course, this decrease of mean velocity is followed by a huge increase in turbulence level. Turbulence intensity is seen to increase fourteen time fold.

Three-dimensional graphs and contour lines for the moments of third and forth order are shown in Figs. 5 and 6.

The major turbulence activities downstream of the keel are observed to be occur in a square box of 3 mm. The implication is that any small imperfections in the model will result in great changes in the flow field. The lack of symmetry in the flow field is seen in Fig. 4 where a shift to the left of about 0.5 mm in the point of highest turbulent intensity is observed. This results from a light misalignment in the support of the keel. This misalignment, unfortunately, could not be removed, for resulting from the construction of the keel. In any case, this only emphasizes how great changes occur in small scales of the flow.

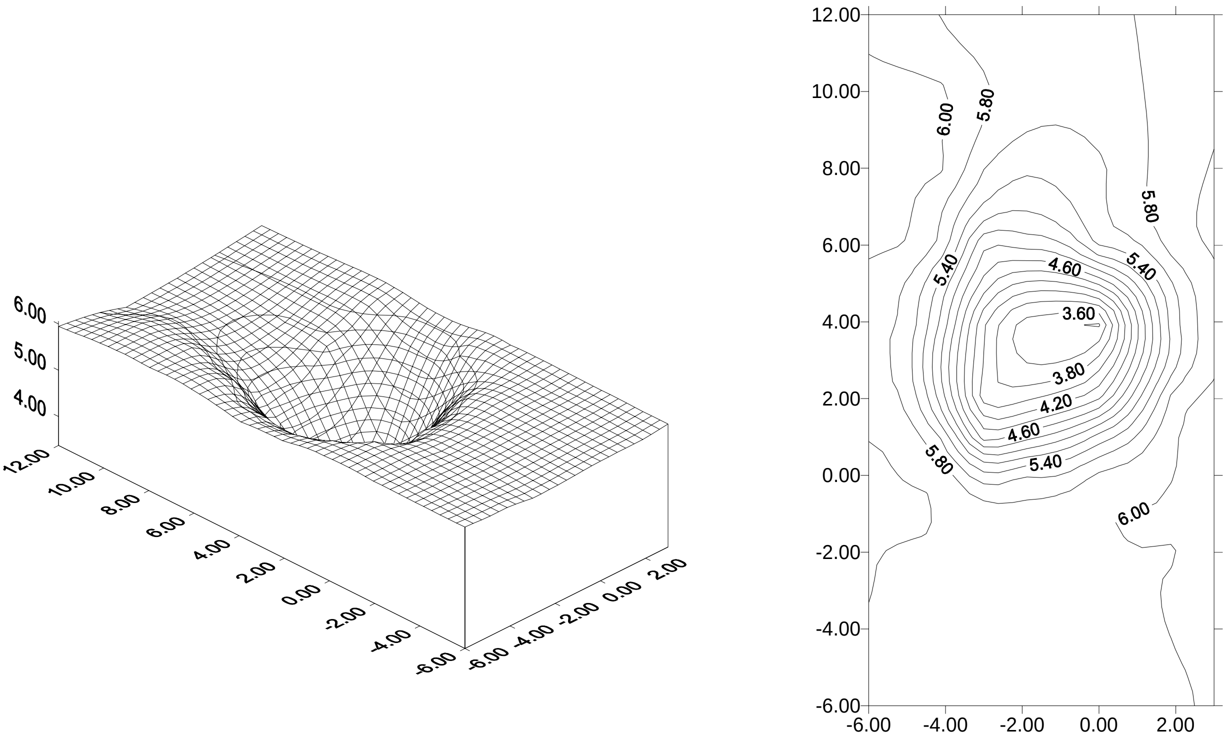

Figure 3. Mean velocity field.

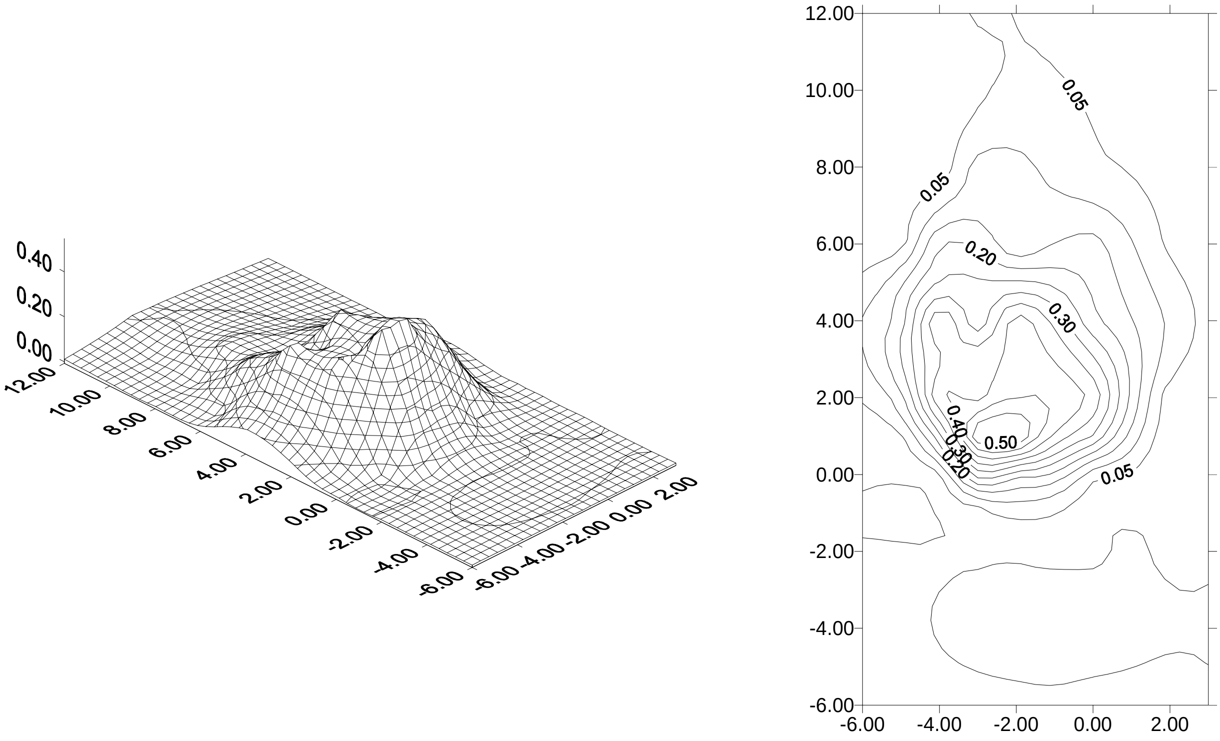

Figure 4. Turbulence second-order moments.

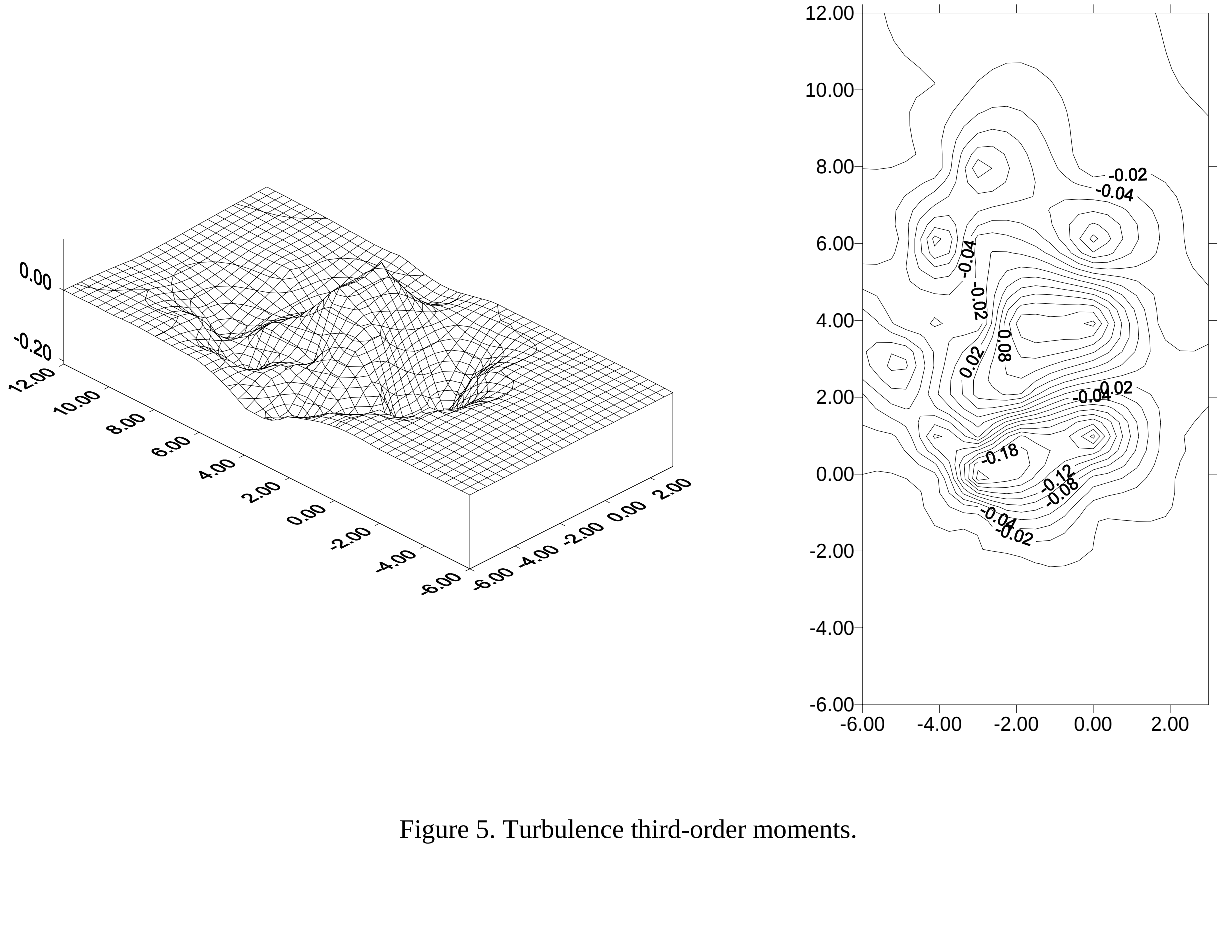

Figure 5. Turbulence third-order moments.

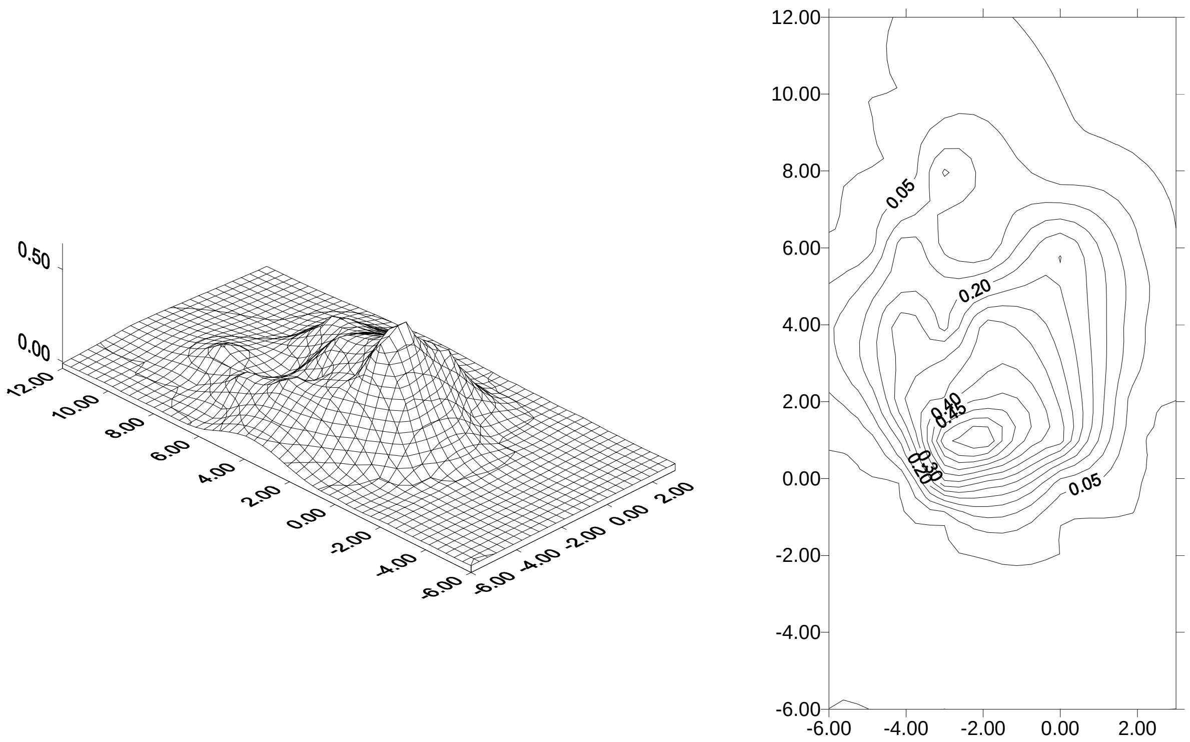

Figure 6. Turbulence forth-order moments.

5. Conclusion

The present work has been very successful in provide valuable data for the validation of numerical codes that aim at furnishing realistic engineering predictions for flows downstream of a keel. In particular measurements for the mean velocity and moments of second, third and forth orders have been provided.

At this point, we find relevant to point out to the reader that the number of such studies in literature is very small. That fact further increases the relevance of this contribution. All data presented here are compared in a companion paper with numerical data obtained from numerical simulations of the flow obtained with eddy-viscosity and Reynolds stress models. As we shall see, the agreement will be within acceptable values.

*Acknowledgements.*APSF is grateful to the Brazilian National Research Council (CNPq) for the award of a research fellowship (Grant No 304919/2003-9). The work was financially supported by CNPq through Grant No 472215/2003-5 and by the Rio de Janeiro Research Foundation (FAPERJ) through Grants E-26/171.198/2003 and E-26/152.368/2002. RMBC benefited from a M.Sc. scholarship from CAPES.

6. References

Barberis, D., Molton, P. and Malaterre, T., 1998, Control of 3D turbulent boundary layer separation caused by a wing-body junction, Exp. Therm. Fluid Sci. AIAA J., 16, 54-63.

Cummings, R. M. and Simpson, R., 1992, Flow past a wing body junction: experimental evaluation of turbulence models, AIAA J., 30, 873-881.

Davenport, W. J., Forsythe, J. R., Morton, S. A. and Squires, K. D, 2003, Computational challenges in high angle of attack flow prediction, Progress Aero. Sciences, 39, 369-384.

Gatski, T. B.and Jongen, T., 2000, Nonlinear eddy viscosity and algebraic stress models for solving complex turbulen flows, Progress. Aero. Sciences, 36, 655-682.

Simpson, R., 2001, Junction Flows, Annu. Rev. Fluid Mechanics, 33, 415-443.