

# DIRECT NUMERICAL SIMULATION OF THE TURBULENT FLOW AROUND AN AIRFOIL USING SPECTRAL/HP METHOD

Bernhard Stoevesandt\*, Andrei Shishkin<sup>†</sup>, Joachim Peinke\*, Claus Wagner<sup>†</sup>

\*Oldenburg University, Hydrodynamics Group  
Carl-von-Ossietzky-Str. 9-11, 26129 Oldenburg, Germany  
e-mail: [bernhard.stoevesandt@uni-oldenburg.de](mailto:bernhard.stoevesandt@uni-oldenburg.de)  
web page: <http://www.forwind.de>

<sup>†</sup>DLR Göttingen,  
Bunsenstr. 10 37073 Göttingen, Germany  
e-mail: [andrei.shishkin@dlr.de](mailto:andrei.shishkin@dlr.de)

**Key words:** Fluid Dynamics, Spectral/hp, DNS, Airfoil

**Abstract.** *Since the accuracy of the widely used Reynolds Averaged Navier Stokes (RANS) approach is limited simulating turbulent flow separation on airfoils, we applied a spectral/hp element method to predict the turbulent flow separation at an airfoil for an angle of attack of  $12^\circ$ . The spectral/hp [1] approach allows to solve the incompressible Navier-Stokes equation with spectral accuracy on unstructured grids by means of Direct Numerical simulation (DNS). Here we present first results we obtained from DNS of the flow around a fx79-w151a airfoil section for a low Reynold Number  $Re = 5 \cdot 10^3$ .*

## 1 INTRODUCTION

The technology of wind turbines used for the generation of sustainable energy has made a lot of progress in the last 10 years. Yet, due to the strong effect of turbulence in the wind field, predicting the aerodynamic characteristics of wind turbines is remains challenging. Sudden changes in the angle of attack caused by atmospheric turbulences lead to dynamic stall. The numerical simulation of aerodynamic loads under dynamic stall conditions is still difficult. Therefore manufacturers of wind turbines rely on estimations if the maximum loads on turbine airfoils have to be calculated.

Numerical simulations based on the solution of the RANS equation rely on turbulence models, which often have difficulties in predicting accurately all required details of the turbulent effects [3]. On the other hand Large Eddy Simulations (LES) and Direct Numerical Simulations (DNS) have become more interesting during the last years as computational power is increasing. Using DNS for airfoil simulations is - due to the still high computational costs at higher Reynolds numbers - not very practical. Some groups involved in

the research of computational fluid dynamics (CFD) have therefore started to apply LES codes for airfoil flow problems [4][5]. The results have been mostly promising, yet predicting accurately all turbulent details on airfoils remains a difficult topic. In this respect the LESFOIL project [5] showed, that predicting the leading edge and trailing edge flow correctly is difficult and strongly depends on the mesh resolution and its topology. While the simulation at high Reynolds numbers has been difficult, some groups have turned to simulate the flow for lower Reynolds numbers to start off with [4][7][8]. Though at lower Reynolds numbers DNS which are known to be more accurate, can be performed for comparison.

In the present work flow simulations at low Reynolds number were performed by means of DNS and will be compared to the results of the corresponding LES in the future (for the LES results see [9]).

Facing these tasks we employed a spectral/hp code to simulate the turbulent flow around an airfoil [1][10]. The advantage of this code is, that it allows a great flexibility in solving the flow around complex geometries with a minimum of costly remeshing. The first results obtained with the DNS solver are analysed.

## 2 NUMERICAL METHOD

As we are considering airfoils for the use on wind turbines and therefore low velocities, it is sufficient to consider incompressible Newtonian fluid flow, which can be described by the dimensionless Navier-Stokes equations:

$$\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla) \mathbf{u} = -\nabla p + \nu \Delta \mathbf{u} + \mathbf{F} \quad (1)$$

$$\nabla \cdot \mathbf{u} = 0 \quad (2)$$

where  $\mathbf{u}$  denotes the velocity,  $p$  the pressure,  $\mathbf{F}$  the forces and  $\nu$  the kinematic viscosity. The velocity  $\mathbf{u} |_{\partial\Omega} = 0$  at on the airfoil.

The spectral/hp method is an attempt to combine the flexibility of finite element or volume methods with the accuracy of spectral methods[1]. The domain is divided into an unstructured grid that can be adapted to the problems faced. For each of the elements  $E$  the spectral method is being applied. A polynomial basis  $\Phi_{lm}(r, s)$  is defined to approximate the function  $f(r, s)$  for each element  $E$  with  $(r, s) \in E$ :

$$u(r, s) = \sum_l \sum_m \bar{u}_{lm} \Phi_{lm}(r, s) \quad (3)$$

Where  $\bar{u}_{lm}$  are the expansion coefficients for the polynomial  $\Phi$  and  $(r, s)$  are local coordinates the Element  $E$ . This way the polynomial order to calculate every element of the grid can be changed. Jacobi polynomials  $P$  are used for the expansion  $P$ . So a triangular orthogonal expansion basis for  $(r, s) \in E = \{-1 \leq r, s; r + s \leq 0\}$  can be setup as

$$\Phi_{lm}(r, s) = P_l^{0,0} \left( 2 \frac{(1+r)}{(1-s)} - 1 \right) (1-s)^l P_m^{2l+1,0}(s) \quad (4)$$

To evaluate the calculations a transformation from triangular to rectangular space is being performed [6][11].

The spectral/hp code  $\mathcal{N}\varepsilon\kappa\mathcal{T}\alpha r$  was used in the current project. The code is parallelized using MPI.

### 3 SIMULATION PARAMETERS

The effects of dynamic stall are investigated on a fx79-w151a profile mainly developed for the use on wind turbines. In spanwise direction the profile is assumed to be homogeneous. The flow around this profile is being investigated under submission of a shortly disturbed laminar inflow at a Reynolds number of  $Re = 5000$  in respect to the chord length  $L_c$  at an angle of attack of  $\alpha = 12^\circ$ . In Fig. 1 a plot of iso surfaces of the velocity in z-direction are presented in order to give an impression of the dimensions of the blade segment.

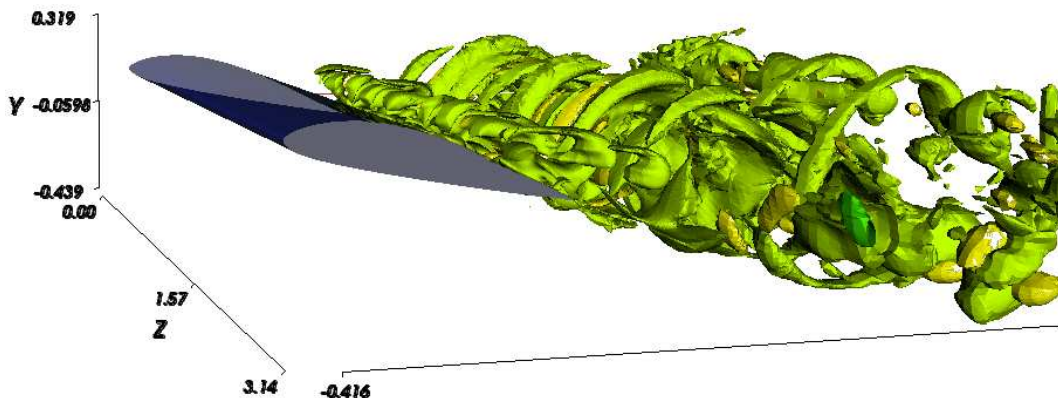


Figure 1: Contour-Plot of the velocity in z-direction on the fx79-w151a airfoil. The units on the axis show are given in chord length  $L_c$ .

The computational domain expands from  $-15L_c$  in the inflow to  $30L_c$  in the outflow direction and from  $-15L_c$  to  $15L_c$  in cross-flow direction. Periodic boundary condition were chosen for the outer boundary of the domain in cross-flow direction. A Fourier expansion was used along the homogeneous spanwise direction of length  $\pi L_c$ .

The simulation was performed on a grid using  $K=1954$  elements. Fig.2 shows a 'z-slice' of the domain and some detailed sections of the mesh around the airfoil as well as the leading and trailing edge. The mesh was generated using the mesh generator gmsh [12]. In the vicinity of the trailing edge the resolution of the grid was increased in this region. To

further improve the resolution in the vicinity of the wall an additional layer of elements was introduced with a mesh size at the wall of  $\Delta y^+ \leq 7.2$ . Such a  $\Delta y^+$  value seems rather coarse for a DNS, but it cannot be compared to a  $\Delta y^+$  value used in a finite volume method, since the accuracy in spectral/hp does not only depend on the mesh size, but also on the polynomial order. Around the airfoil the polynomial order used was  $P = 9$ . As the error decreases exponentially with an increase of the polynomial order (see [1]), the effective  $\Delta y^+$  is  $\leq 1$ .

Finally, in the spanwise direction, 64 Fourier planes were used in our DNS.

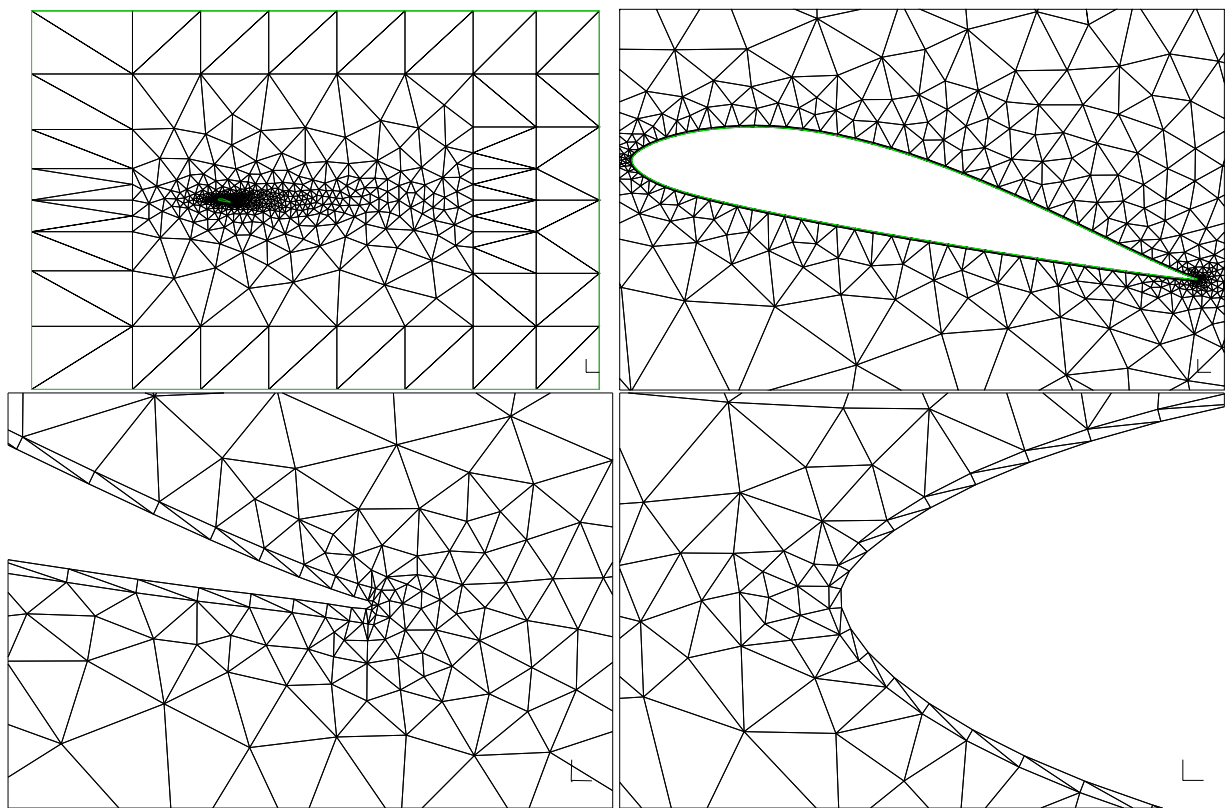


Figure 2: Grid in the (x,y)-plane consisting of 1954 elements (top left). The density of elements was strongly increased at tip and tail of the airfoil.

At the beginning of our DNS the polynomial order  $P$  was set to 5. Together with the number of elements and Fourier planes, this lead to about 1300000 degrees of freedom, which was still to low for a well resolved DNS (compare e.g. [10]). As the number of elements was already quite large for the spectral/hp method, the polynomial order was varied in the domain for the simulations. Tab.1 reflects the areas of the different polynomial orders and the number of elements calculated using these orders. This approach increased the number of degrees of freedom to about 3570000. The density of degrees of

freedom per volume units varies strongly from the order between  $10^7$  to  $10^5$  at the foil to the order of 1 on the outer area of the domain.

Since the polynomial order was varied, also the time stepping was varied during the simulation between time steps of  $0.5 \cdot 10^5$  and  $2.5 \cdot 10^5$  in dimensionless time units. This way the CFL value was reaching 0.35 in a maximum.

Area	Top right point of area	Low left point of area	P	Number of Elements
Rest Domain	(30,15)	(-15,-15)	5	690
Foils surrounding	(1.1,0.4)	(-0.4,-0.25)	9	1003
Near wake	(5.7,1)	(1.1,-0.6)	7	261
Airfoil	(0.76, -0.16)	(-0.22,0.046)	0	0

Table 1: Areas of certain polynomial orders in respect to airfoil position.

### 3.1 Computational Performance

The computations were performed using 16 processors on the CLUH-Cluster of the RRZN Hannover a 16 MEGWARE Saxonid C4 Compute-Knot Quad-Processor-System and on a 64 bit Opteron Linux-Ethernet Cluster at the University of Oldenburg. In Fig.3 the speed-up obtained at the CLUH-cluster is presented. The linear speed-up underlines the good parallel performance of the used code.

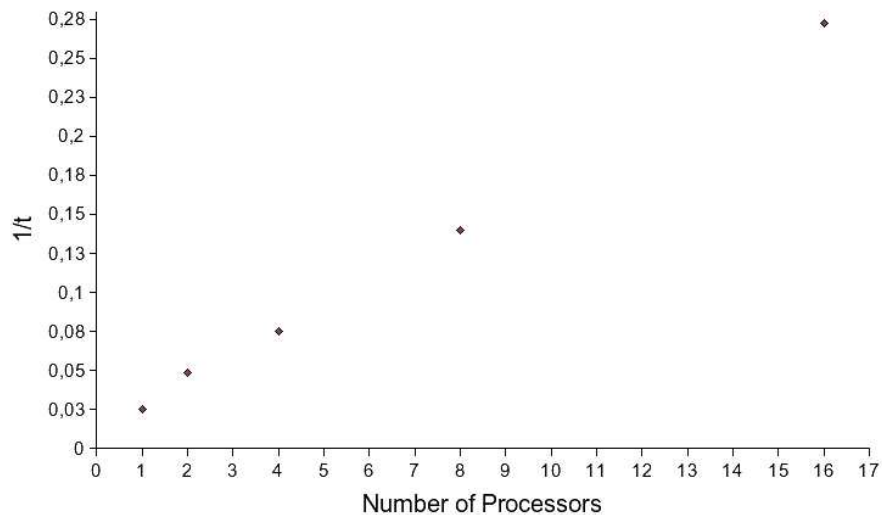


Figure 3: Needed averaged inverse computational time per time step over the number of processors. Doubling the number of processors from 8 to 16 still increases the performance by 95%.

## 4 FIRST RESULTS

The forces in  $\mathcal{N}_{\varepsilon\kappa\mathcal{T}\alpha r}$  are calculated as

$$F = - \int_s P n ds + \int_s \tau \cdot n ds. \quad (5)$$

The plots in Fig. 4 display the drag and lift forces over a short period of time. It shows the response of the lift and drag to the turbulent flow separation at the blade. So far the average values of the lift and drag coefficient are  $C_l = 0.77$  and  $C_d = 0.2$ . In Fig.

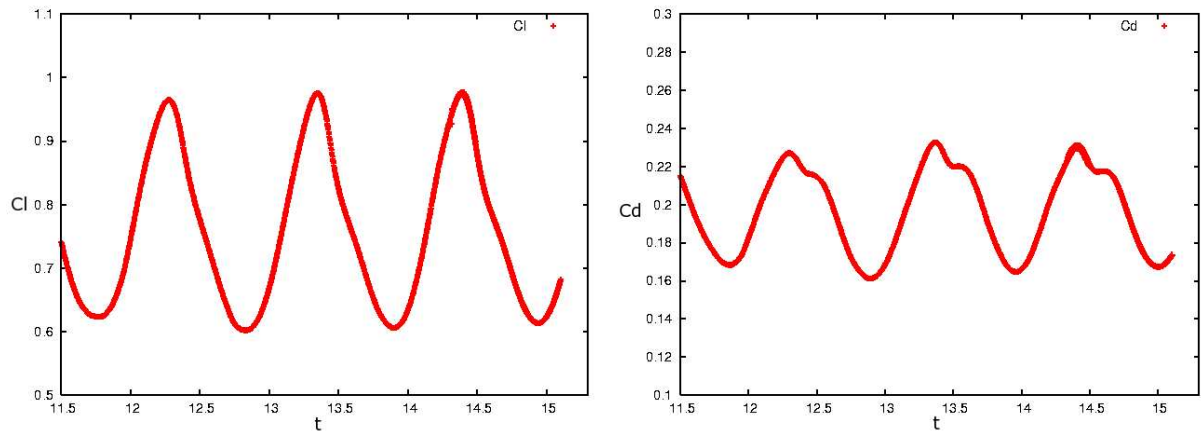


Figure 4: Lift (left) and drag (right) for the airfoil over dimensionless time at  $Re = 5 \cdot 10^3$  and  $\alpha = 12^\circ$ . As the Reynolds number is low, the fluctuations are slow.

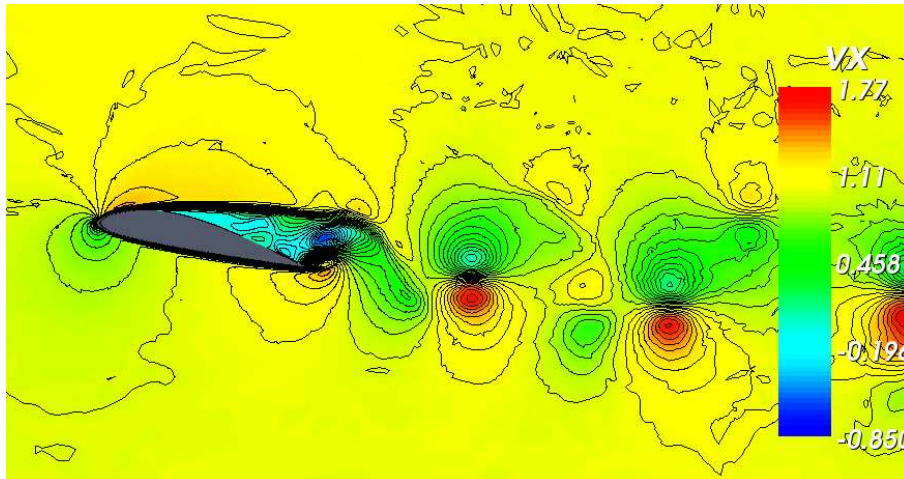


Figure 5: Contours of u-velocity in a (x,y)-plane cut.

5 and 6 contours of the u- and v-velocity are presented respectively at a (x,y)-plane cut. Flow separation can be observed in the region of  $x/L_c = 0.3$ . Fig. 7 and 8 show a course

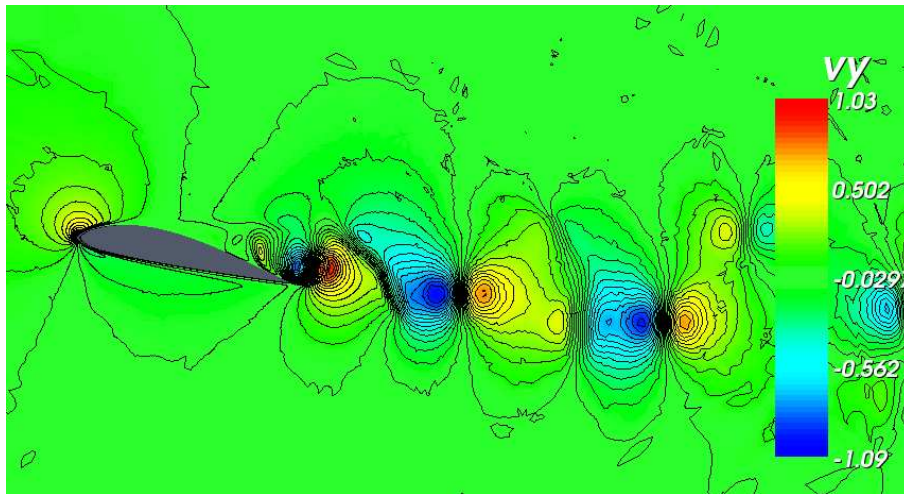


Figure 6: Contours of v-velocity in a (x,y)-plane cut.

of the vortice separation at the trailing edge by pressure contours. In order to perform a detailed statistical evaluation of the flow field, we are continuing the computation in order to present the results at the conference.

## 5 CONCLUSIONS

Using the spectral/hp method is due to its flexibility very practical for simulating turbulent flow in complex domains. Especially the possibility of locally increasing the accuracy by increasing the polynomial order in certain grid elements without the need of remeshing has been very useful. A mesh consisting of few elements can thus still be used for high accuracy calculations. Thus it seems to be possible to increase the Reynolds number for upcoming DNS of the flow around the airfoil.

The first results show the turbulent flow and a small flow separation region at the trailing edge. A verification of the simulation is to be done in the future. So far there are no comparable measurements at  $Re = 5000$ . The computations have to be continued for some time in order to obtain profound statistical results from the flow fields. Additionally the statistical variation of the turbulent wind field shows a non Gaussian distribution in wind speed and direction[2]. So the next goal will be to resolve the statistical variation of loads on the blade caused by such wind fields. Nevertheless the first results look promising for the future work to use spectral/hp methods as a tool to investigate on dynamical stall effects on rotor blades of wind turbines.

## 6 ACKNOWLEDGEMENTS

We are grateful to Prof. S. Sherwin for providing the  $\mathcal{N}\epsilon\kappa\mathcal{T}\alpha r$  code we used for our work.

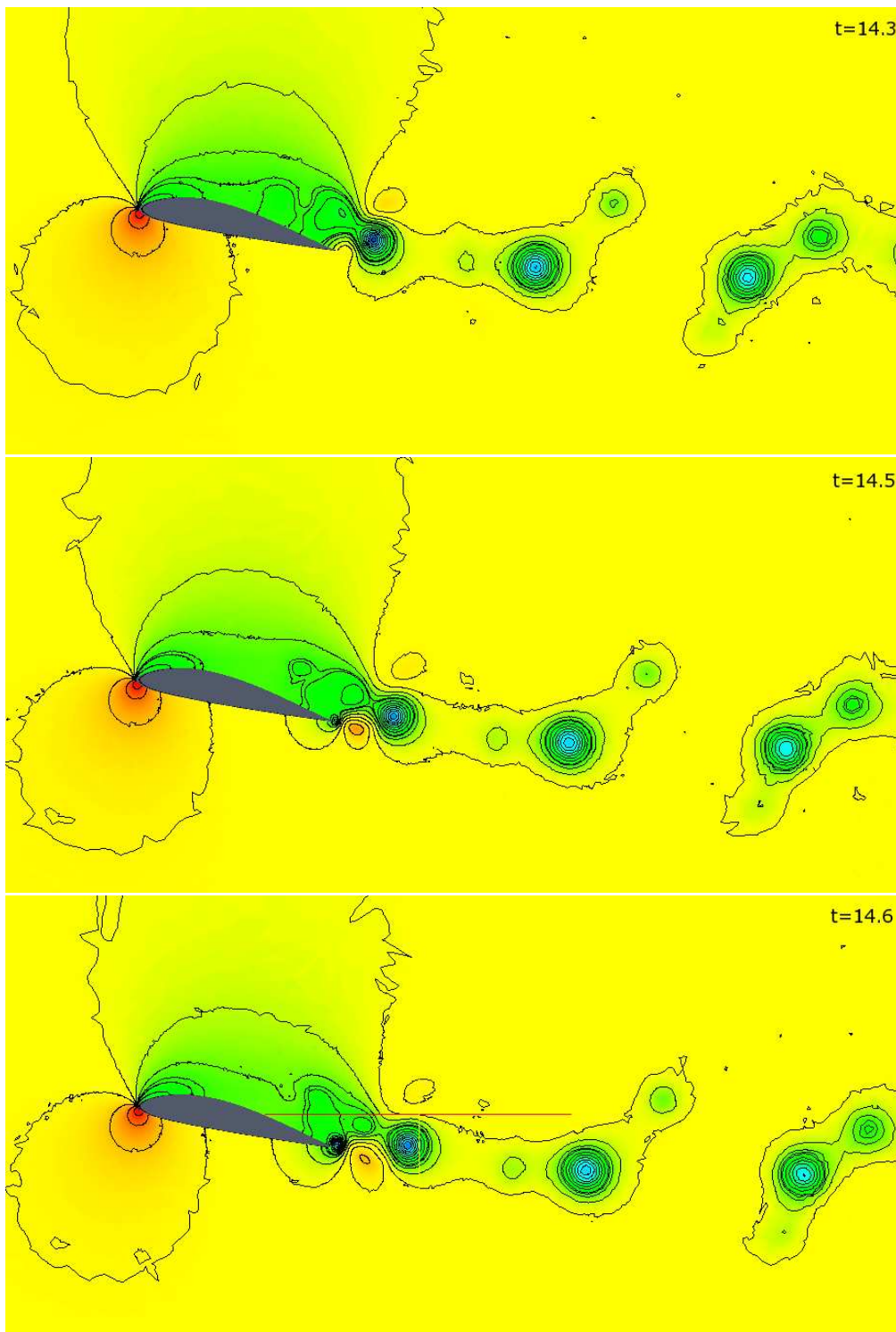


Figure 7: Contours of the pressure over a time period for  $Re = 5 \cdot 10^3$ .

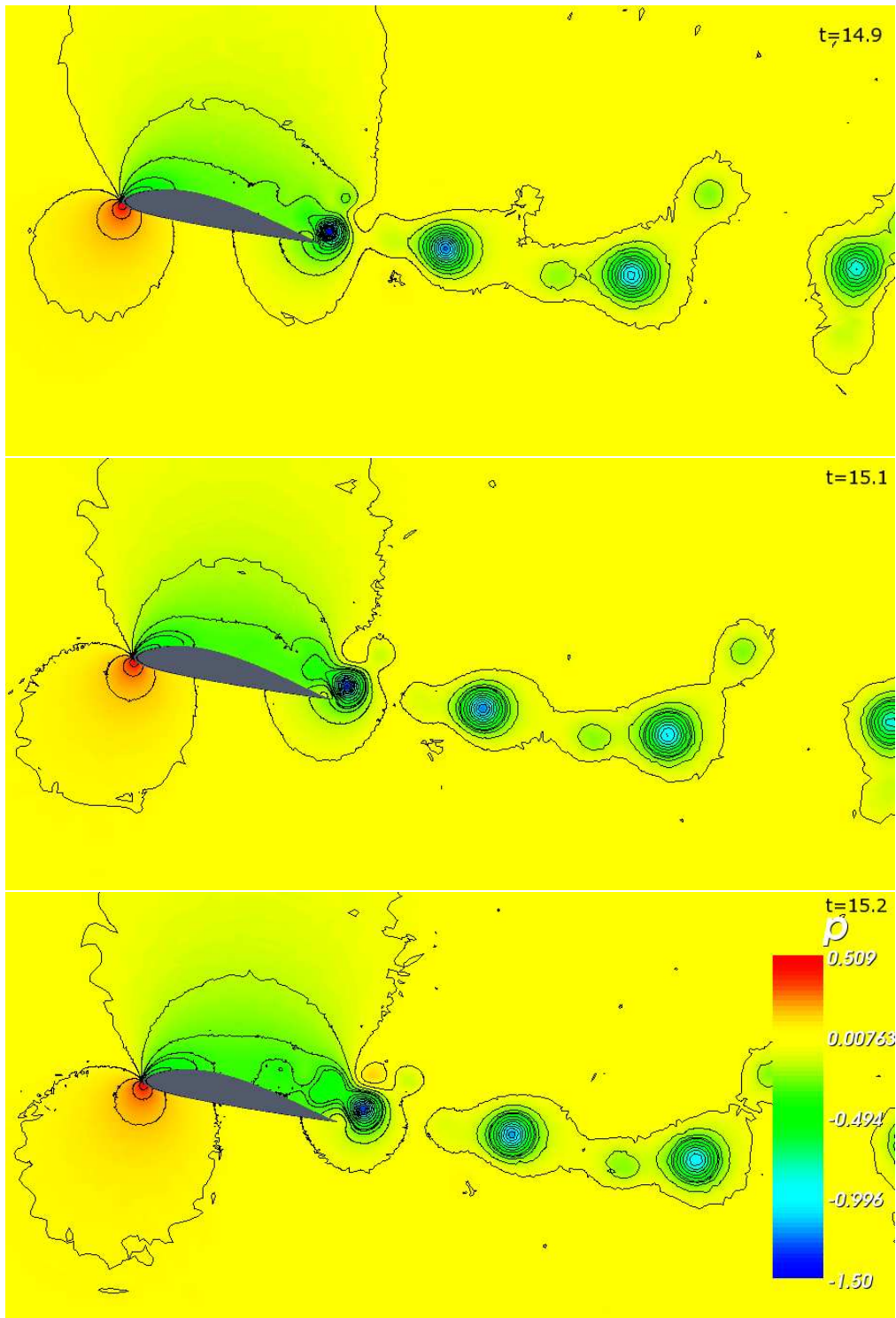


Figure 8: Contours of the pressure over a time period for  $Re = 5 \cdot 10^3$ .

## REFERENCES

- [1] G. E. Karniadakis and S. J. Sherwin *Spectral/HP Element Methods for CFD* Oxford University Press, Oxford, 1999
- [2] F. Boetcher *Statistische Analyse der atmosphärischen Turbulenz und allgemeiner stochastischer Prozesse*, Ph. D. - Thesis p.72 , University of Oldenburg, 2005
- [3] S. Dahlström and L. Davidson: "Large Eddy Simulation of the Flow Around an Aerospatiale A-aerofoil", *ECCOMAS 2000, European Congress on Computational Methods in Applied Sciences and Engineering*, 11-14 September, Barcelona, Spain, 2000.
- [4] F. Richez , V. Gleize , I. Mary and C. Basdevant: Zonal RANS/LES coupling simulation of a transitional and separated flow around an airfoil near stall, *Conference on Turbulence and Interactions TI2006*, Porquerolles, France, 2006
- [5] L. Davidson, D. Cokljat, J. Fröhlich, M. A. Leschziner, C. Mellen, W. Rodi (Eds.): *LESFOIL Large Eddy Simulation of Flow Around a High Lift Airfoil*, Notes un Numer. Fluid Mech., vol. 83, Springer, 2003
- [6] T. Warburton *Spectral/hp element methods on polymorphic domains*, PhD Thesis, Brown University, 1998
- [7] M. Breuer, N. Jovicic: An LES Investigation of the Separated Flow Past An Airfoil at High Angle of Attack, *Proc. of the 4th Workshop on DNS and LES, Enschede Netherlands*, ERCOFTAC Series, vol.8, pp. 165-172, Direct and Large Eddy Simulation, Kluwer Academic Publ., Dordrecht, 2001
- [8] N. Jovičić, M. Breuer, J. Jovanović: Anisotropy-Invariant Mapping of Turbulence in a Flow Past an Unswept Airfoil at High Angle of Attack, *J. Fluids Engineering*, vol. 128(3), pp. 559–567, (2006).
- [9] A. Shishkin and C. Wagner: Large Eddy Simulation of the Flow Around Wind Turbine Blade, *to be published in the Proc. of the European Congress on Computational Methods in Applied Sciences and Engineering (ECCOMAS2006)*, Eindhoven, Netherlands, 2006.
- [10] X. Ma, G. S. Karamanos, G. E. Karniadakis : Dynamics and low-dimensionality of a turbulent near wake, *J. Fluid Mech.*, vol. 410, pp. 29-65, Cambridge University Press, 2000
- [11] G. S. Karniadakis, S. J. Sherwin, R. M. Kirby, X. Wan: Spectral/hp Element Methods for Deterministic and Stochastic Flow Problems, Preprint submitted to Elsevier Science, 2005

- [12] C. Geuzaine and J.-F. Remacle. Gmsh: a finite element mesh generator with built-in pre- and post-processing facilities. <http://www.geuz.org/gmsh/>, 2006