Journal of Physics: Conference Series

2D Numerical Investigation of the Laminar and Turbulent Flow Over Different Airfoils Using OpenFOAM

To cite this article: H Rahimi *et al* 2014 *J. Phys.: Conf. Ser.* **555** 012070

2D Numerical Investigation of the Laminar and Turbulent Flow Over Different Airfoils Using OpenFOAM.

H.Rahimi[1], W.Medjroubi[2], B.Stoevesandt[1] and J.Peinke[1,2]

[1] Fraunhofer IWES, Ammerländer Heerstr. 136, 26129 Oldenburg, Germany
[2] ForWind, Ammerländer Heerstr. 136, 26129 Oldenburg, Germany

E-mail: `hamid.rahimi@uni-oldenburg.de`, `w.medjroubi@uni-oldenburg.de`

Abstract. The aim of this work is to assess the prediction capabilities of the turbulence models and the transition model kkl-ω available in OpenFOAM and to achieve a database of airfoil aerodynamical characteristics. The airfoils chosen for the simulations are **FX 79-W-15A** and **NACA 63-430**, which are widely used in wind turbines. The numerically obtained lift and drag coefficients are compared with available experimental results. A quantitative and qualitative study is conducted to determine the influence of meshing strategies, computational time step together with interpolation and temporal schemes. Two Reynolds Averaged Navier-Stokes models (RANS models) are used, which are the k-ω SST model by Menter and the kkl-ω model (which involves transition modeling) by Walters and Davor.

1. Introduction

Nowadays, wind energy is a very important source of renewable and clean energy. Optimizing the energy output of wind turbine has caused the increase in wind turbines size, which in return caused their power capacity to increase by a factor of 100 [3]. In order to achieve the maximum energy output, both structural and aerodynamical properties of wind turbine blades are decisive and have to be improved. The optimal blade design with regards to wind speed leads to lower costs and maximizes torque and lift over drag ratio. CFD codes, which deliver reliable computational results and can predict the aerodynamical properties of airfoils are needed for the optimization of the wind turbine airfoils shape. In this context, thick and flatback airfoils can offer advantages for the rotor root design. These airfoils are desirable from a structural point of view, as there thickness and their blunt trailing-edge increases the sectional area and the momentum of inertia. This effect induces a reduction of overall blade weight and an increase in its structural strength [8]. Moreover, these types of airfoils are insensitive to roughness effects and provide a high lift coefficient [4]. Nevertheless, there is a drag penalty at the trailing-edge of flatback airfoils, this penalty is not important when considering inboard sections. Adding to the high drag generated at the trailing-edge of flatback airfoils, a higher noise level was registered for these airfoils when compared to sharp edge airfoils [1]. Our goal in this investigation is to use the Open Source Field Operation and manipulation package (OpenFOAM) [7] to simulate the turbulent flow field over wind turbines airfoils using different turbulence models. The airfoils chosen for the simulation are FX 79-W-151A and NACA 63-430. This choice is motivated by two main reasons, first these airfoils are widely used in wind turbines and secondly experimental data for these airfoils is available [9, 2].

The comparison between our simulation results and experimental and other numerical results will contribute to set up a best practice guide for simulating the flow over airfoils using OpenFOAM and to validate the simulation against available results from the literature. In Section 2 of this investigation, the numerical method used in OpenFOAM as well as the computational settings for two different RANS models are reported. In Section 3 simulation conditions of FX 79-W-151A and NACA 63-430 using the different RANS methods are introduced, and the result in terms of lift, drag and pressure coefficient values are compared with experimental and other computational results. Finally, in Section 4, the conclusions regarding the prediction capabilities of the turbulence models used are presented.

2. Numerical Method

In this section the numerical method implemented in OpenFOAM is introduced. This includes a short description of OpenFOAM along with the turbulence models used. A brief overview concerning the numerical mesh used for the simulations is also presented.

2.1. OpenFOAM

OpenFOAM is an open source CFD package written in $C++$, and it consists of modifiable libraries which contain different numerical models and CFD tools. OpenFOAM uses Finite Volume Method (FVM), where separated matrix equations are created for each equation and solved using iterative solvers. The method uses a co-located strategy, which means that the solution variables for each matrix equation are defined at cell centers. The FVM in OpenFOAM is applied to arbitrary shaped cells on unstructured grids. The coupling between pressure and velocity equation is performed using Pressure-Implicit Split-Operator algorithm (PISO) for transient simulations and the Semi-Implicit Method for Pressure-Linked Equations (SIMPLE) algorithm for steady simulations. In OpenFOAM, a wide variety of turbulence models are implemented. They range from RANS models to Large Eddy Simulation (LES) models including hybrid methods as Detached Eddy Simulation (DES).

2.2. Turbulence Models

In the present work two RANS models are used:

- **k-ω SST Model** is a two-equation turbulence model by Menter [5], which solves two transport equation for the kinetic energy k and the specific dissipation ω.
- **kkl-ω Model** is a transition model based on k-ω SST model for incompressible flows, in which a third transport equation is solved to predict the magnitude of low frequency velocity fluctuations that trigger transition in the boundary-layer [6]. In this model, the transition is not fixed but triggered by the velocity fluctuations.

Other turbulence model, were used (DES and RANS (Spalart-Allmaras)) but not presented here.

2.3. Mesh and boundary conditions

OpenFOAM possesses a built-in tool to create structured meshes, called blockMesh [7] . Among various kinds of meshes which one can use for simulating the flow over an airfoil, a C-Mesh is used in this investigation. This mesh represents some advantages as it is simple and easy to create using blockMesh and it is ideal for 2D airfoils with a sharp or a blunt trailing edge. In Fig. 1 an example of the C-Mesh used in this investigation is shown, and a view of the computational mesh is presented in Fig. 2. In order to improve the stability of the numerical simulations, the outer boundary is located far away from the airfoil, at 25 chord lengths. A denser mesh is used in the regions of interest such as the boundary-layer and the airfoil near-wake regions. The boundary conditions used during all simulations are as follow: on the airfoil no slip conditions are used for the velocity ($U = V = 0$) and Neumann boundary conditions for the pressure. On the right of the numerical domain outflow conditions are imposed, which are Neumann boundary condition for the velocity and zero pressure ($p = 0$). On the rest of the domain boundaries inflow conditions are considered ($U = U_\infty$) and Neumann boundary condition for the pressure, where U_∞ is the freestream velocity.

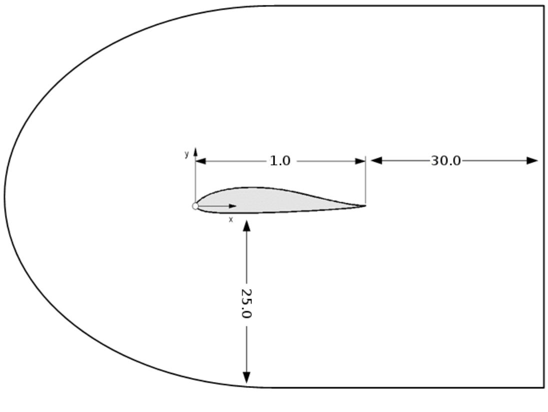


Figure 1. Schematic representation of the numerical domain used in this investigation. The dimensions are written in terms of the airfoil chord length c.

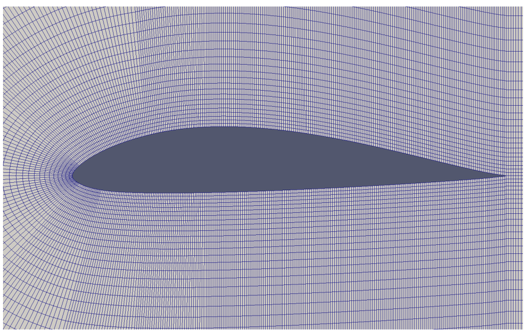


Figure 2. Close-up of the grid around the airfoil FX 79-W-151A. Mesh is shown for every 4^{th} cell.

Since different turbulence models are used in this work, different types of meshes with respect to the height of first cell and aspect ratio are used. For k-ω SST, a wall function is used and the height of the first cell was adjusted to obtain a $y^+ \geq 200$ (where y^+ is the non- dimensional wall distance). For kkl-ω model the height of the first cell was adjusted to obtain $y^+ \simeq 1$, as in this model no wall functions are used in the boundary-layer region. The k-ω SST model was used for steady (not presented here) and unsteady simulations. The transition model, kkl-ω, was only used in steady mode, due to time limitations.

3. Simulations and Results

In this section the numerical simulations set-up is introduced. The available experimental data for the lift coefficient values are compared with the numerical simulation obtained using OpenFOAM for different angles of attack, using the above mentioned turbulence models. Pressure coefficient distribution along the airfoil are also plotted, and when possible, compared to available experimental results.

3.1. Results for NACA 63-430

This airfoil belongs to the NACA family and has 30% thickness. The experimental data [2], which is based on experiments conducted in the VELUX wind tunnel are compared with the numerical results obtained using OpenFOAM. The Reynold number is 1.5×10^6, corresponding to an inflow velocity of 24m/s, and a turbulence intensity of 1% is considered. The mesh properties and simulation initial condition are listed in tables 1 and 2.

Table 1. Simulation condition for k-ω SST model for NACA 63-430.

Element	value/Style
Total number of cells	598,500
yPlus	($y^+ \simeq 200$)
Reynolds number	1.5×10^6
Freestream velocity	24 m/s
Solver	k-ω SST
	Unsteady

Table 2. Simulation condition for kkl-ω model for NACA 63-430.

Element	value/Style
Total number of cells	934,425
yPlus	($y^+ \simeq 1$)
Reynolds number	1.5×10^6
Freestream velocity	24 m/s
Solver	kkl-ω
	Steady-simpleFoam

When using the k-ω SST turbulence model, a fully turbulent flow is assumed, i.e. no transition modelling is applied. In this investigation, the k-ω SST turbulence model is used with wall function in the boundary-layer region. In Fig. 3, for most angles of attack (AOA), the values of the lift coefficient obtained from the numerical simulations are in very good agreement with experimental results [2]. Due to the long simulation time, only stall and post-stall simulations were conducted using the k-ω SST model. When using the transition model kkl-ω with a grid satisfying $y^+ \simeq 1$, the model is able to correctly predict the lift coefficient values. It is also important to note that the results obtained using kkl-ω transition model are in better agreement than the ones obtained using the Re_θ transition model [9] as presented in reference [2]. In terms of drag, numerical data were only available for simulations conducted with the kkl-ω transition model and are plotted in Fig. 4. The transition model is able to correctly predict the drag coefficient values for all considered angles of attack.

In fig. 5 the Cp curve for kkl-ω transition model and experimental results are presented for angles of attack in the pre-, stall and post-stall regimes. It is important to note that, the transition model is able to capture the real physics of the flow, as a very good agreement between the numerical and experimental C_p curves is found here. However, there are some discrepancies at $AOA = 10°$, around the stall angle, at the trailing-edge, where the transition model is unable to predict the pressure curves. This can be attributed to the fact that, around the stall angle, the flow is highly unsteady. Another source for the discrepancies in the CFD results can be the high aspect ratio cells present in the numerical grid. These cells are caused by the restriction $y^+ = 1$ for the transition model combined with the need to obtain a numerical grid with a reasonable total number of cells.

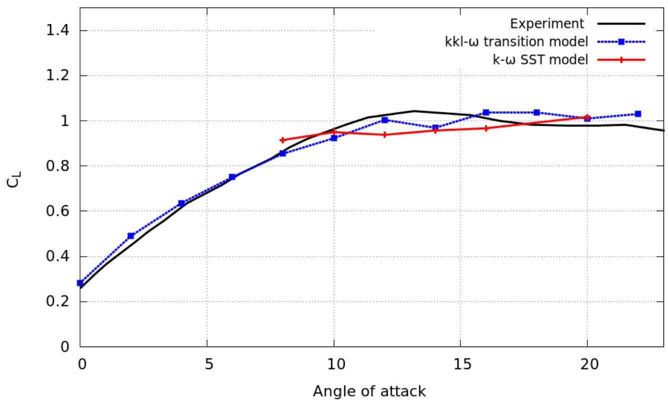


Figure 3. The lift coefficients for different angles of attack for the airfoil NACA 63-430. The experimental results are taken from [2].

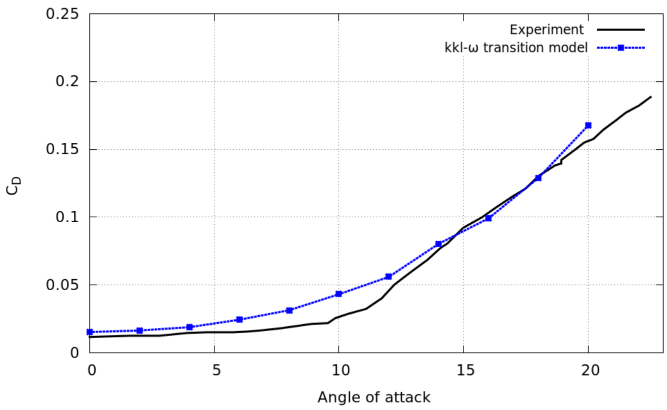
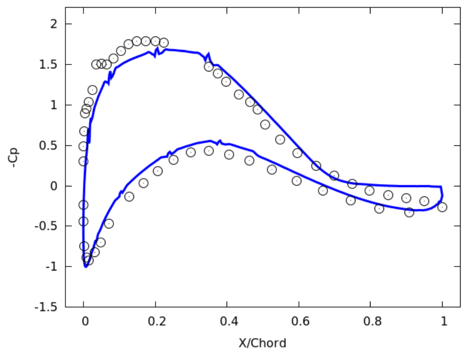


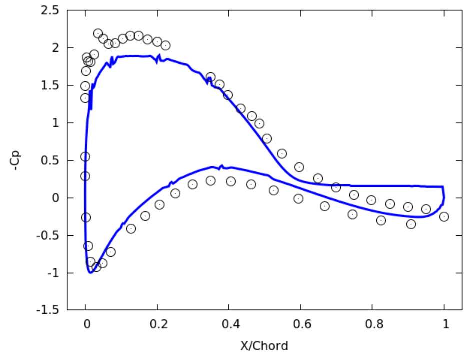
Figure 4. The drag coefficients for different angles of attack for the airfoil NACA 63-430. The experimental results are taken from [2].

3.2. Results for FX 79-W-151A

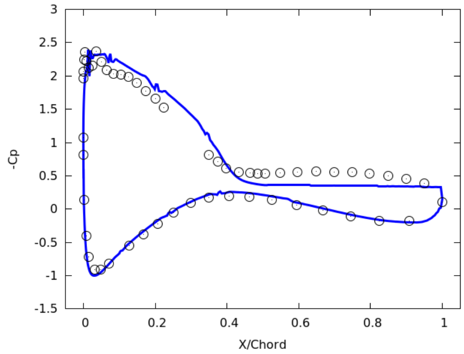
The FX 79-W-151A is a commonly used airfoil in wind turbines root sections. The simulations for this profile have been conducted for Re=7×10^5 for which experimental results are available



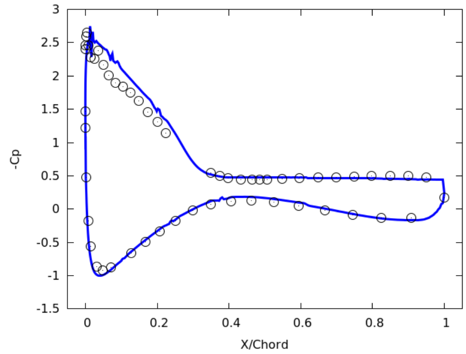
(a) $AOA = 6°$



(b) $AOA = 10°$



(c) $AOA = 16°$



(d) $AOA = 20°$

Figure 5. Pressure coefficient distributions for the NACA 63-430 airfoil at (a) $AOA = 6°$, (b) $AOA = 10°$, (c) $AOA = 16°$, and (d) $AOA = 20°$. The open circles represent the measurements, the blue line represents the computational results obtained with the kkl-ω transition model.

[9]. The inflow velocity is 10.5 m/s and a turbulence intensity of 1% is considered. The mesh and simulation initial conditions are shown in table 3 and 4 for both turbulence models used.

Table 3. Simulation condition for k-ω SST model for FX 79-W-151A.

Element	value/Style
Total number of cells	947,450
yPlus	($y^+ \simeq 200$)
Reynolds number	7×10^5
Freestream velocity	10.5 m/s
Solver	k-ω SST
	Unsteady

Computational simulations where conducted using k-ω SST turbulence model in a fully turbulent configuration. In addition, calculations are performed using the transition model kkl-ω turbulence model.

When using the k-ω SST model, see Fig. 6, the results in terms of lift coefficient values are qualitatively and qualitatively in good agreement for all angles of attack. Although, a slight over prediction of C_L is observed at the highest angles of attack $AOA > 20°$. When using

Table 4. Simulation condition for kkl-ω model for FX 79-W-151A.

Element	value/Style
Total number of cells	1507,650
yPlus	$(y^+ \simeq 1)$
Reynolds number	7×10^5
Freestream velocity	10.5 m/s
Solver	kkl-ω
	Steady-simpleFoam

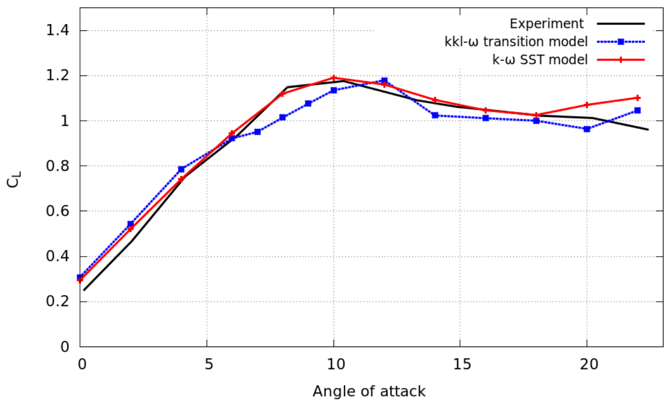


Figure 6. The lift coefficients for different angles of attack for the airfoil FX 79-W-151A. The experimental results are taken from [9].

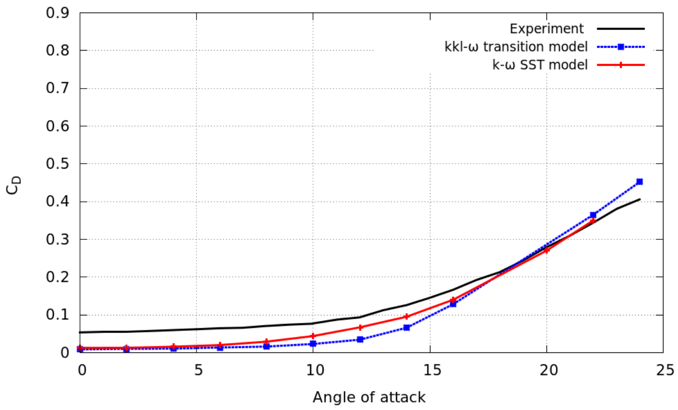
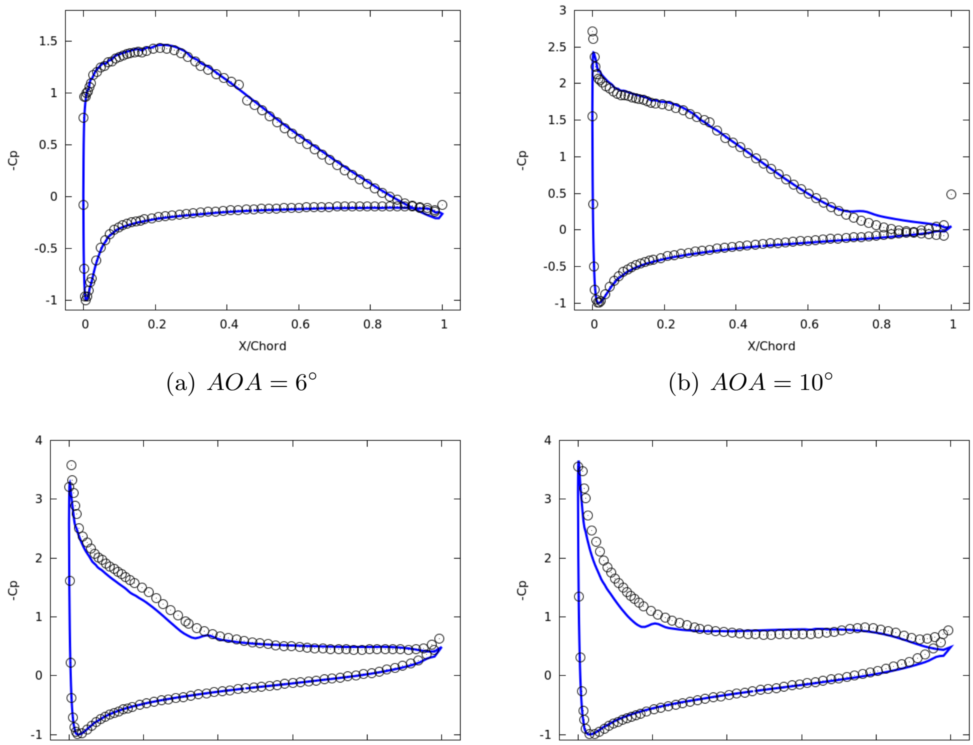


Figure 7. The drag coefficients for different angles of attack for the airfoil FX 79-W-151A. The experimental results are taken from [9].

the transition model, a reasonable agreement is found between our results and the experimental results [9]. In the region $6^\circ < AOA < 10^\circ$, the transition model under predicts the lift coefficient. This can also be due to grid effects, especially the dependence of the results on the y^+ as mentioned above for the NACA 63-430 airfoil. Nevertheless, the transition model is able to reproduce the experimental results with a reasonable degree of agreement.



(a) $AOA = 6°$ (b) $AOA = 10°$

(c) $AOA = 16°$ (d) $AOA = 20°$

Figure 8. Pressure coefficient distributions for the FX 79-W-151A airfoil at (a) $AOA = 6°$, (b) $AOA = 10°$, (c) $AOA = 16°$, and (d) $AOA = 20°$. The open circles represent the kkl-ω transition model, the blue line represents the computational results obtained with the k-ω SST model.

The computational results obtained for the drag using the kkl-ω transition model and using the k-ω SST model are shown on Fig. 7. The k-ω SST model is able to correctly predict the drag coefficient values for all considered angles of attack, but the transition model under predicts the C_D values. This can also be attributed to grid effects mainly caused by the high aspect ratio of the grid cells.

In Fig. 8 the C_p curves at four different angles of attack, are plotted for the results obtained using the transition model and the k-ω SST model. Experimental results were not available for the pressure coefficient distributions. Fig. 8 shows a clear agreement between the results obtained with k-ω SST and the transition model.

4. Conclusions

The viscous and incompressible flow over two types of airfoils is simulated in steady and unsteady configurations, using k-ω SST model and the transition model kkl-ω, which are both implemented in OpenFOAM. The computational results obtained show a qualitatively and quantitatively good agreement for both turbulence models for pre-stall and post-stall regions. The capability of the k-ω SST and the transition model in reproducing lift, drag and pressure coefficients is assessed in this investigation. It is also shown that the transition model, kkl-ω, delivers much better result when compared with the Re_θ transition model, as it is based on a more physical assumption of the transition process. The influence of the grid characteristics on the prediction of the aerodynamic coefficients and the pressure distribution, especially the aspect ratio of the grid cells, should be further investigated. The results obtained in this investigation for 2D cases are encouraging. 3D simulations will be conducted in the future to test the transition model capabilities. This will permit the use of OpenFOAM in building a database for airfoil aerodynamical characteristics to be used in the design of wind turbine airfoils.

References

[1] D. E. Berg and M. Barone. Aerodynamic and aeroacoustic properties of a flatback airfoil (will it rumble or whisper?).

[2] F. Bertagnolio, N. Sørensen, J. Johansen, and P. Fuglsang. Profile catalogue for airfoil sections based on 3d computations, risø-r-1581. Technical report, Risø National Laboratory, Roskilde, Denmark, 2006.

[3] T. Burton, D. Sharpe, N. Jenkins, and E. Bossanyi. *Wind energy handbook*. Wiley, New York, 2001.

[4] M. Fuller. PhD thesis.

[5] F. R. Menter. Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA*, 32:1598–1605, 1994.

[6] F. R. Menter and T Esch. Elements of industrial heat transfer predictions. In *16th Brazilian Congress of Mechanical Engineering*, 2001.

[7] OpenFOAM. The open source computational fluid dynamics toolbox, 2013. `http://www.openfoam.com/`.

[8] SAND2003-0723 Sandia Report. Innovative design approaches for large wind turbine blades. Technical report, Sandia National Laboratories, 2003.

[9] J. Schneemann. Auftriebmessungen in turbulenter umgebung. Master thesis, Institut of Physics, University of Oldenburg, 2009.