High fidelity CFD-CSD aeroelastic analysis of slender bladed horizontal-axis wind turbine

M. Sayed¹, Th. Lutz¹ and E. Krämer¹, Sh. Shayegan ², A. Ghantasala ², R. Wüchner² and K.-U. Bletzinger²,
¹ Institute of Aerodynamics and Gas Dynamics (IAG), University of Stuttgart, Pfaffenwaldring 21, D-70569 Stuttgart
² Chair of Structural Analysis, Technische Universität München (TUM), Arcisstr. 21, D-80333 München
E-mail: sayed@iag.uni-stuttgart.de

Abstract. The aeroelastic response of large multi-megawatt slender horizontal-axis wind turbine blades is investigated by means of a time-accurate CFD-CSD coupling approach. A loose coupling approach is implemented and used to perform the simulations. The block-structured CFD solver FLOWer is utilized to obtain the aerodynamic blade loads based on the time-accurate solution of the unsteady Reynolds-averaged Navier-Stokes equations. The CSD solver Carat++ is applied to acquire the blade elastic deformations based on non-linear beam elements. In this contribution, the presented coupling approach is utilized to study the aeroelastic response of the generic DTU 10MW wind turbine. Moreover, the effect of the coupled results on the wind turbine performance is discussed. The results are compared to the aeroelastic response predicted by FLOWer coupled to the MBS tool SIMPACK as well as the response predicted by SIMPACK coupled to a Blade Element Momentum code for aerodynamic predictions. A comparative study among the different modelling approaches for this coupled problem is discussed to quantify the coupling effects of the structural models on the aeroelastic response.

1. Introduction

Currently, the trend in wind turbine design is up-scaling which results in larger slender and flexible wind turbine blades. Therefore, as wind turbine blades become lighter and more flexible, aeroelastic instabilities are of great concern due to the coupling of the unsteady aerodynamic loads with the elastic deformations [1]. Moreover, the aeroelastic instabilities strongly affect the operational life of the turbines [2]. Hence, aerodynamic-structure coupling analyses are required to predict the complex interaction of the unsteady aerodynamic loads and the elastic deformations of the blades.

Till now, most aeroelastic models are based on simplified aerodynamic models using the Blade-Element Momentum theory (BEM) [3, 4]. The BEM-based method is computationally efficient and provides reasonable estimations of the aeroelastic behaviour of flexible blades, but some aerodynamic phenomena are not captured accurately. Mainly, BEM theory is based on the one-dimensional momentum equilibrium. In fact, it neglects any three-dimensional effects. Therefore several corrections are needed in order to account for all complex unsteady and 3D aerodynamic effects [5]. As the wind turbine rotor diameter increases, including more complex
geometries such as pre-cone, rotor tilt and pre-bended blades, symmetric inflow conditions can not be assumed anymore. Therefore, it becomes inaccurate simulating it with the dynamic inflow model of the BEM based simulations [6].

In principle all occurring viscous, 3D and unsteady aerodynamic effects are captured with CFD. Nevertheless, high-fidelity CFD is much more expensive compared to the traditional BEM methods, but they are capable of resolving the unsteady and complex flow field around the wind turbine including all complex geometries. To overcome the traditional BEM limitations and to improve the aerodynamic prediction accuracy, coupled CFD-CSD methods have been frequently used. Up to now, in the wind energy, high-fidelity aeroelastic investigations have been found little attention, only recently they have come into the focus due to increasing rotor radius. Many of the recent 3D CFD-CSD investigations have been performed on the NREL 5MW rotor [6, 7, 8]. Yu and Kwon [6] investigated the aeroelastic response using a loosely coupled CFD-CSD method solving the incompressible Navier-Stokes equations on unstructured grid for the blade forces and non-linear Euler-Bernoulli beam for the deformations. The interaction between the fluid and the structure model was carried out over every rotor revolution. Bazilevs and Hsu [7] [8] utilized an FSI between a low-order Arbitrary Lagrangian-Eulerian Varational Multi-Scale (ALE-VMS) flow solver and a non-uniform rational basis spline (NURBS) based structural solver.

In the present study, the CFD and CSD solvers and the coupling interface are described in section 2. The full discription of the used wind turbine in this study and the computational setups are then discussed in section 3. Then the aeroelastic response and effects on the blade performance are evaluated in section 4. The results are compared with the CFD results by modelling the blade as a rigid model to investigate the effects of blade deformations on the rotor blade aerodynamic performance.

2. Numerical Solvers

2.1. CFD Solver
FLOWer is the CFD solver utilized in the present study which it is provided by the German Aerospace Center DLR [9] with specific extensions implemented at the IAG to obtain the aerodynamic loads by solving the Reynolds-averaged Navier-Stokes equations (RANS). Different turbulence models are available in FLOWer and the Wilcox k-ω turbulence model [10] with fully turbulent flow state is used. Implicit dual time-stepping scheme is utilized for the time integration of the RANS equations according to Jameson [11]. An Arbitrary Lagrangian Eulerian (ALE) approach is employed for the grid motions of the rotating parts. The grid over-set method is applied by means of the CHIMERA technique which enables better meshing control of complex geometries.

In order to enable coupling FLOWer to other structure solver and to achieve the proposed CFD-CSD coupled computations, two different modules are needed as shown in Figure 1. The first module is the surface pressure load integration module which is used to calculate the forces needed by the CSD solver. The second is the mesh deformation module needed to deform the flexible parts to given deformations at every time-step. The deformation module implemented at the IAG is based on a Radial Basis Functions (RBF). Separate object oriented C++ libraries are used for the implementation of both modules [12]. Parallel computing is enabled to increase the efficiency of the computation by using a multi-block structures of the grid.

In recent years, a process chain for wind turbine CFD simulations was developed at the IAG to accelerate the pre-processing stage of simulations. This process chain includes a high quality blade grid generation script and one must carefully set the different parameters to control and insure the quality of the generated mesh. Several CFD simulations of different offshore and onshore wind turbines have been conducted using that process chain at IAG [13, 14, 15].
2.2. CSD Solver
The CSD solver used to solve the structure dynamics is Carat++. It is a general finite element solver developed over the last years at the Chair of Structural Analysis TUM. Different elements can be used such as spring, damper, mass, truss, beam, membrane, shell and solid elements including geometrical and material non-linearity as well as some composite material description for membranes and shells. It is used to perform different structural analyses as follows:

- Static (linear/nonlinear).
- Dynamic (linear/nonlinear).
- Eigenfrequency.

In order to first verify the structural model of the blade, eigenfrequency and nonlinear static analyses (using a reference load case) are carried out. During the CFD/CSD coupling, the blade’s structural model is solved using a nonlinear dynamic analysis. Implicit generalized alpha time stepping scheme is used to discretize the time. A Rayleigh damping model is used to model the damping properties of the blade. The coefficients of the Rayleigh damping are derived from the logarithmic damping (3%) and natural frequencies of the first flapwise and edgewise modes of the blade provided in [17].

2.3. CFD-CSD Coupling Interface
The coupling framework used in this study is the Enhanced Multi-Physics Interface Research Engine (EMPIRE) developed at Technical University of Munich (TUM) [16]. It can be used for co-simulation with multiple-codes which suits the multi-physics problems. EMPIRE can be used for the following tasks as described in Figure 2.

- Data communication among multiple-codes.
- Set up co-simulation flexibly with different coupling algorithms.
- Mapping between non-matching grids.
- Different coupling algorithms and coupling control.
- Surface reconstruction for dimensionally reduced structural models.

There is a data communicator in EMPIRE called Emperor. It is used as an MPI server-clients communicator to receive/send data from/to different clients. However, each code is defined as a client to be used as a storage of interface data inside Emperor. Each client can have one/multiple predefined meshes, data fields on these meshes and/or signals. At the beginning of the communication, Emperor allocates memory for these data. During co-simulations, Emperor can be used for mapping between the different meshes from different clients, do relaxation to...
control the coupling stability and do extrapolation at the beginning of a new time-step to improve the stability of the iterative coupling providing a better initial guess.

There is no specific coupling interface used in FLOWer, therefore, coupling FLOWer to EMPIRE will allow further improved aero-elastic results by using different structure models. Another advantage of coupling FLOWer to EMPIRE is that EMPIRE is already coupled to Carat++. Loose coupling is implemented and used where the data between the CFD solver and the CSD solver are transferred at time-step basis. The coupling scheme implemented in this work is shown in Figure 3.

3. Computational Setups

3.1. Wind Turbine Configuration

The wind turbine examined in the present study is the generic reference turbine of the ongoing European InnWind project [17]. It is a large HAWT with 178.3 m rotor diameter, a hub height of 119 m and FFA-W3 airfoil series. The rotor is installed at 5° tilt angle and a cone angle of -2.5°. Due to the rotational periodicity of the 3-bladed rotor, a 120 degree sector of the rotor is considered in the present study. Therefore, only a single blade and one-third of nacelle/hub are modelled where the aerodynamics of the full rotor are taken into account through periodic constraints. Figure 4 shows an adequate geometrical CAD surface model of one blade and nacelle/hub established using CATIA which is a multi-platform CAD/CAM/CAE commercial software developed by the French company Dassault Systemes. The wind turbine blade and
nacelle/hub rotate clockwise as viewed from the inlet boundary (x-direction). Pre-bended blades are used with about 3.73%R at the blade tip. The blade at the root region is equipped with wedge shaped Gurney flaps to increase the aerodynamic performance [17].

3.2. CFD Setup

Gridgen® is used in this research to generate the volume meshes for all sub-domains. For each sub-domain, an independent grid is created with adopted refinement. The final generated CFD volume meshes are based on a structured overset mesh technique. This overset feature allows more mesh control over all the wind turbine parts, as every component is meshed separately. Finally the volume domains are placed together and overlapped using CHIMERA overlapping mesh technique. The blade mesh is generated and refined by a special scripts developed at IAG. A background mesh is created to extend the computational domain to the far field [14].

The blade volume mesh (the red in Figure 6) is generated (blade sectional mesh is shown in Figure 7) using the proven script at the IAG [13, 14]. 300 cells are used in chord-wise direction and 100 cells in (span-wise) radial direction. 35 cells are introduced to resolve the boundary layer with $y^+ \approx 1$ for the first boundary layer cells.
The nacelle/hub mesh (the green in Figure 6) is also generated taking into account the $y^+ \leq 1$ condition at the wall and also the no-slip boundary condition is applied. The cell size at the surface in the axial direction is 0.07 m at the hub and is 0.3 m. The nacelle/hub is connected to the blade by means of a rotating blade-hub-connector mesh. The background domain (the gray one in Figure 6) is created using a one third cylinder with a periodic boundary condition. The far field dimensions are defined in terms of the blade radius (R) as follows: 6R in the upstream direction, 9R in the downstream direction and 6R in the far field direction. These values are chosen based on studies done by Sayed et. al. [14]. The final grid statistics of the different volume domains used in the CFD and the coupled simulations are shown in Table 1.

| Domain             | Blade | Hub and Nacelle | Background | Blade-Hub-Connector | Total number of cells |
|--------------------|-------|-----------------|------------|---------------------|----------------------|
| Number of cells    | 5.2   | 2.4             | 5          | 0.5                 | 13.1                 |
| [millions]         |       |                 |            |                     |                      |

### 3.3. CSD Setup

While by using shell or solid elements the local structural behavior of the blade’s surface (e.g. local stresses and buckling) can be captured more precisely, the global behavior of the blade and its effect on the surrounding flow is more important for evaluating the wind turbine’s performance in an FSI simulation.

The global behaviour of the blade can be captured using beam elements, with the main advantage of having considerably lower computational cost compared to shell or solid elements. Therefore, in the present study the blade’s structure is modelled using 51 3D non-linear co-rotational beam elements which allow large rotations. The beam formulation is based on the Timoshenko Beam theory which takes into account the shear deformation. Each beam element has 6 Degree of Freedom (DOF) per node (3 translational and 3 rotational DOFs). In [17] the cross section stiffness and mass properties of predefined sections along the radius of the blade are given. The section stiffness properties (bending, shear, torsional and extensional stiffness) are given with respect to the principal axes of each section with the origin at the elastic center of the section. In the blade’s structural model used in the current study, the beam nodes pass through the shear centers of the predefined sections and the beam properties are defined in a global coordinate system with the origin at the blade’s root and the Z-axis in the radial direction of the blade. Therefore a series of transformations was done to transform the local section properties provided in [17] to Global values used in CARAT++. Figure 8 shows the
predefined cross sections (in green) together with the beam finite element mesh of the blade (in black).

4. Results

Initially, for the rigid wind turbine blade, CFD-only analyses are performed for steady inflow conditions. Afterwards, an unsteady simulation for the wind speed of 11 m/sec is performed. This simulation is used as a reference for the CFD-CSD simulations and for validation purposes. In this simulation the rotor shaft tilt angle is ignored for one-third model. CFD validations were performed by [14] and showed a good agreement to other published literature. Hereafter, a modal analysis of the blade is utilized for structure validation purposes. Then CFD-CSD simulations are conducted by the implemented FLOWer-EMPIRE-Carat++ coupling with flexible blade in the CFD solver. The effect of coupled simulations on the blade performance is studied. Moreover, the effects of the blade pre-cone angle and the gravitational and centrifugal forces on the blade deformations are examined. Finally, the results are compared to two different coupled results: first, to the CFD-CSD loose coupling using FLOWer as flow solver and SIMPACK as structure solver and second, to the BEM-CSD loose coupling results.

4.1. CSD Validation

First, a frequency analysis of the blade is carried out using CARAT++. The results are compared with the reference values available in [17]. The eigenfrequency values are listed in Table 2 and show a good agreement with the reference values.

| Mode          | Natural frequency [Hz] |
|---------------|------------------------|
| 1st flap mode | 0.63                   |
| 1st edge mode | 0.87                   |
| 2nd flap mode | 1.74                   |
| 2nd edge mode | 2.73                   |

In the next step, the response of the structure under a simple reference load case is simulated in a nonlinear static analysis. Constant point loads with specific magnitudes are applied on certain nodes of the beams and the tip displacement of the blade is measured. Figure 9 compares the flapwise deflection of the blade from CARAT++ with the values from [18] calculated by research groups at the Technical University of Denmark (DTU) and the National Renewable Energy Center of Spain (CENER). The results show a good agreement with the reference values.

![Figure 9: Normalized flap-wise deflection under simple reference load case [18]](image-url)
4.2. Aeroelastic Response

For the one-third model, steady flow simulations for the rigid blade are performed to get converged results to be used as a starting point for the coupled simulations. Then, CFD-CSD simulations are conducted at a time-step corresponding to 1° azimuth. The pre-cone angle is set to zero and three revolutions are simulated to achieve convergence. The elapsed time for this simulation is approximately 46 hours using 420 processors 2.6 GHz CPUs. All the coupled FLOWer-EMPIRE Carat simulations are conducted at the SuperMUC cluster at the Leibniz-Rechenzentrum.

As shown in Figure 10, the flap-wise (out of plane), edge-wise (in-plane) and torsion blade-tip deformations are normalized by the blade radius and presented over three revolutions. As shown the magnitude of the deformations increases over the first 45° azimuth before it starts to oscillate around the mean value. The flap-wise deformations converge almost after the first revolution whereas the edge-wise and torsion deformations need almost two revolutions to approach the converged values. The reason for that is the unsteady nature of the flow over the rotating blade and the dynamics of the blade structure. After the convergence has been shown for CFD (order of $10^{-6}$), CSD (order of $10^{-6}$) and aero-elastic results (converged blade tip deformations), the spanwise distribution of the blade deformation is presented in Figure 11. It can be observed that the blade deformations are quite large. The tip blade-bending deflections are 0.1033R and -0.0186R in the flap-wise and edge-wise directions respectively.

![Figure 10: Azimuthal variations of the normalized blade-tip deformations](image)

![Figure 11: Normalized blade deformations after 3 revolutions](image)

4.3. Aeroelastic Effects on the Blade Performance

To investigate the effect of these large deformations on the wind turbine performance, the azimuthal variations of the power and thrust predicted by the coupled CFD-CSD over the last revolution are evaluated. The mean power and thrust values are calculated by the integration over the last revolution of the CFD-only simulations. Then the CFD-only and the coupled CFD-CSD results are normalized by these mean values and presented in Figure 12. As shown, the rotor power and thrust are reduced by approximately 1% and 0.5% respectively. The reason
for that is the reduction in the local angle of attack due to the nose down torsion deformation which exhibits almost $0.2^\circ$ (nose down) at the blade tip as shown in Figure 11c.

![Figure 12: CFD-CSD performance results compared to CFD-only for the last revolution](image)

In order to investigate further the effect of the blade deformations on the performance, the blade sectional loads in the flap-wise direction, edge-wise direction and the twisting moment are used as shown in Figure 13. For the range between the blade root and $r/R = 0.2$ the differences in all load cases between the CFD-only and the coupled CFD-CSD is due to the unsteady nature of the flow at the hub region. Sayed [14] has discussed that for the case of CFD-only simulations. From $r/R = 0.2$ to $r/R = 0.35$ there is almost no difference due to the small deformations as a consequence of large blade rigidity in this region. Very little increase can be seen in the flap-wise and edge-wise load for the region of $r/R = 0.4$ and $r/R = 0.7$. This is because a slight increase in the local angle of attack as shown in Figure 14a. Finally, in the outer part between $r/R = 0.7$ and $r/R \approx 0.95$, the coupled forces are reduced due to the angle of attack reduction which leads to a reduction in the leading edge suction pressure as shown in Figure 14b and 14c. Conversely, a significant increase in the blade twisting moment can be noticed, as shown in Figure 13c, which is expected to occur due to the large edge-wise deflection with slight decrease in the edge-wise force.

![Figure 13: Blade loads distributions](image)

### 4.4. Effect of The Pre-cone Angle

For slender bladed rotors large deformations are expected, therefore, to avoid collision between the tower and the blades, greater clearance distance between the tower and the rotor is needed. Tilt angle, pre-cone angle and pre-bended blades are used for that purpose. As the simulations in this work deal with a one-third model, the use of the tilt angle is not applicable. Therefore, in addition to the pre-bended blades, a pre-cone angle of $-2.5^\circ$ is applied. The effect of the pre-cone angle on the coupled results is presented and discussed in terms of the blade tip deformations over
the last revolution. In addition, the results are compared to these without pre-cone. Applying a pre-cone angle reduces the rotor swept area and as a consequence decrease the power of the turbine. The power reduction is due to load reduction in the edge-wise direction and hence the edge-wise deformation also reduced as shown in Figure 15b. This reduced edge-wise deformation is a consequence of decreasing the local angle of attack due to the nose down twist, therefore augmentation of the flap-wise deformation is expected.

4.5. Effect of Gravitational and Centrifugal Forces
In addition to the effect of the aerodynamic loads on the coupled CFD-CSD results, the impact of the gravitational and centrifugal forces ($F_g$ and $F_c$) is discussed. For large bladed wind turbines, it is expected that these forces will have great influence on the blade deformations. The simulations have been conducted for the same configurations with pre-cone angle of -2.5°. The blade tip deformations over the last revolution are presented in Figure 16. It can be concluded that $F_g$ and $F_c$ increase the deformation amplitude in all directions and reduce the mean flap-wise deformation. Also there are small fluctuations during the oscillations due to structure dynamics. To find the source of these fluctuations, CSD-only dynamic simulation under the effect of $F_g$ and $F_c$ only is performed. The predicted tip-deformations are presented in Figure 17. It can be concluded that the blade experiences non-negligible deformations in all directions under the influence of these forces. The maximum edge-wise deflection always at azimuth angles of 90° and 270° in opposite direction. Furthermore, it is concluded that additional coupled revolutions are needed or CSD-only dynamic simulations should be conducted before the coupled CFD-CSD simulations are started.
4.6. Effect of Structural Models

In order to validate the new implemented CFD-CSD coupling, comparisons to the results from an other achieved coupling at IAG to an other structure model (SIMPACK) are presented. The blade is discretized by means of a Multi-Body System (MBS) and the same CFD setup in FLOWer is used in this simulation. The blade deformations at the tip at 360° azimuth angle are presented in Figure 18 and show good agreement with the newly implemented coupling in this paper. The blade is modelled using 51 non-linear Timoshenko flexible beam elements, each element has six degrees of freedom per node (3 translations and 3 rotations). Therefore no big differences are expected between the structural dynamic response in Carat and SIMPACK.

5. Conclusion

In the present scheme, a dedicated CFD flow solver is coupled to different structural models by a loose-coupling. The aeroelastic response of the generic DTU 10MW reference rotor subjected
to rated uniform wind speed was calculated. It is found that the blade aerodynamic loads are reduced due to the aeroelastic effects. The influence of the pre-cone angle on the deformations is discussed and it is found that with pre-cone angle the flap-wise and torsion deformations are augmented whereas the edge-wise deformation decreased. Moreover, the effect of the gravitational and centrifugal forces on the aeroelastic response are discussed and showing only slight influence on the results. Results are compared to BEM based aeroelastic simulations. In addition, the results by implemented coupling approach is compared to the aeroelastic results predicted by FLOWer coupled to MBS tool to evaluate the influence of the structural models on the aeroelastic response.

Acknowledgments
The DAAD (Deutscher Akademischer Austauschdienst) and IGSSE TUM (International Graduate School of Science and Engineering) are acknowledged for financial support. Moreover, the authors would like to acknowledge the colleagues at IAG and SWE of University of Stuttgart for the knowledge shared during the BMWi funded Kontest project.

References
[1] Ahlström A. Influence of wind turbine flexibility on loads and power production. Wind Energy. 2006;9(3):237–249.
[2] Hansen MOL, Soens JN, Voutsinas S, Sorensen N, Madsen HA. State of the Art in Wind Turbine Aerodynamics and Aeroelasticity. Progress in Aerospace Sciences. 2006;42(4):285–330.
[3] Riziotis VA, Voutsinas SG, Politis ES, Chaviaropoulos PK, Hansen A, Madsen Aagaard H, et al. Identification of structural non-linearities due to large deflections on a 5MW wind turbine blade. European Wind Energy Conference and Exhibition; 2008. p. 9–14.
[4] Jeong M, Yoo S, Lee I. Aeroelastic analysis for large wind turbine rotor blades. AIAA; 2011. p. 9–14.
[5] Madsen HA, Riziotis V, Zahl F, Hansen MOL, Snell H, Grasso F, et al. Blade element momentum modeling of inflow with shear in comparison with advanced model results. Wind Energy. 2012;15(1):63–81.
[6] Yu DO, Kwon OJ. Time-accurate aeroelastic simulations of a wind turbine in yaw and shear using a coupled CFD-CSD method. Journal of Physics Conference Series. 2014 Jun;524(1):012046.
[7] Bazilevs Y, Hsu M, Kiendl J, Wüchner R, Bletzinger KU. 3D simulation of wind turbine rotors at full scale. Part II: fluid-structure interaction modeling with composite blades. International Journal of Numerical Methods in Fluids. 2011;65(13):236–253.
[8] Hsu MC, Bazilevs Y. Fluid-structure interaction modeling of wind turbines: simulating the full machine. Computational Mechanics. 2012;50(6):821–833.
[9] Kroll N, Rosso C, Becker K, Rhieme F. The MEGAFLOW project. Aerospace Science and Technology. 2000;4:223–237.
[10] Wilcox D. Multiscale model for turbulent flows. AIAA Journal. 1988;26(11):1311–1320.
[11] Jameson A. Time dependent calculations using multigrid, with applications to unsteady flows past airfoils and wings. In: AIAA 10th Computational Fluid Dynamics Conference; 1991.
[12] Kranzinger PP, Kowarsch U, Schuff M, Keler M, Krämer E. In: Advances in Parallelization and High-Fidelity Simulation of Helicopter Phenomena. Springer; 2015. p. 479–494.
[13] Schulz C, Klein L, Weihing P, Lutz T, Krämer E. CFD studies on wind turbines in complex terrain under atmospheric inflow conditions. In: Journal of Physics: Conference Series. vol. 524. IOP Publishing; 2014. p. 012134.
[14] Sayed M, Lutz T, Krämer E. In: Aeroelastic analysis for large wind turbine rotor blades. CRC Press; 2015. p. 773–780.
[15] Weihing P, Meister K, Schulz C, Lutz T, Krämer E. CFD Simulations on Interference Effects between Offshore Wind Turbines. Journal of Physics: Conference Series. 2014;524(1):012143.
[16] Sicklinger S, Belsky V, Engelmann B, Elmqvist H, Olsson H, Wüchner R, et al. Interface Jacobian-based co-simulation. International Journal of Numerical Methods in Engineering. 2014;98(6):418–444.
[17] Christian B, Frederik Z, Robert B, Taeseong K, Anders Y, Henriksen L, et al. Description of the DTU 10 MW Reference Wind Turbine; 2013.
[18] Lekou DJ, Chortis D. Results of the benchmark for blade structural models:Part A. Innwind deliverable report 2.21; 2013.