University of Brescia

Doctoral Research in Mechanical and Industrial Engineering
"Energy, Fluid & Thermal, and Manufacturing Systems and Technologies"

Unsteady simulation of the incompressible turbulent flow through a vertical axis wind turbine with a high-order Discontinuous Galerkin solver

Tutor:
Ing. Antonio Ghidoni

PhD student:
Gianmaria Noventa

Abstract: In recent years Discontinuous Galerkin (*DG*) methods emerged as one of the most promising technique for time accurate computations, such as Direct Numerical Simulations (*DNS*), Large Eddy Simulation (*LES*) and *hybrid RANS-LES* methods. During this project the high-order DG code MIGALE will be extended for the unsteady incompressible turbulent 3D numerical simulations of a vertical axis wind turbine (*VAWT*). Two high-order temporal schemes will be implemented with adaptive control of the time step, namely a Runge-Kutta and a Rosenbrock-type method. In order to increase the computational efficiency of the solver different techniques will be investigated: (i) pseudo-compressibility method and dual-time stepping technique coupled with the Runghe-Kutta scheme; (ii) different strategies of preconditioning. *X-LES* method will be also implemented to investigated differences in the VAWT simulation with respect to standard RANS approaches.

KEY WORDS: *CFD*, *DG*, unsteady simulation, *RANS* equations, turbulent flows, *hybrid RANS-LES* methods, *VAWT*, incompressible flow, Runge-Kutta schemes, preconditioning, adaptive time step.

Background

> *"Sustainable energy is the sustainable provision of sustainable energy development that meets the needs of the present without compromising the ability of future generations to meet their needs."*

The wind energy is an example of sustainable energy: mechanical energy or electrical energy can be obtained through the conversion of the kinetic energy of the wind. In the past, wind energy was used directly as motive energy, for example in windmills. Today, apart from specific applications, the public interest has focused on its immediate conversion into electricity by means of wind turbines.

In 2010 the production of wind power has been more than 2.5% of all global electricity consumption, with an estimated growth of 25% per annum. The prediction says that the fraction of the global energy demand covered by wind may increase by one percentage point every three years [10]. Wind turbine companies, that produce technologies suited to the exploitation of this energy source, pushing strongly towards research, with the aim of enhancing, optimizing the existing technologies and improve their versatility, so as to allow a greater global spread.

Besides the more traditional horizontal axis wind turbine ($HAWT$), also the turbines with vertical axis ($VAWT$) are carving out a increasing space. A $VAWT$ is characterized by a reduced amount of movable parts in its structure, which gives a high resistance, and an easier maintenance. In fact most of the mechanical components are positioned on the ground. The main advantage of this type of turbines is the ability to harness the wind in any direction, ensuring greater versatility than the horizontal axis wind turbine. This model has also some drawback, particularly the lower efficiency compared

to *HAWT*, due to the engine torque, which is not uniform with the height from the ground and with the angle of rotation.
The design of a wind turbine need a careful study of the blades aerodynamic, to optimize the arrangement of the profile and the selection of the operating parameters.
During the last two decades, the considerable advances in algorithm development and the huge increase of computer power have made *CFD* a key discipline in this industry. The wind turbine community has started to look at *CFD* codes to complement wind tunnel and in field tests to better understand the complex flow physics around rotating wind turbine blades.

The turbulent and unsteady flow through a *VAWT* can be numerically investigate following different approaches: *LES*, *RANS* and *hybrid RANS-LES*. The use of the Reynolds-Averaged Navier-Stokes (*RANS*) equations allows to model all scales by means of a turbulence model. Although the solution of *RANS* equations is feasible even for high Reynolds numbers, it can be inaccurate in the prediction of some flow features such as massive separation laminar recirculation bubbles and transition.
Large Eddy Simulation (*LES*) solves the large scales of turbulence and modelling the effects of smaller scales by means of sub-grid scale (*SGS*) models. The scale separation is obtained by applying a filter to the governing equations which also influences the form of the *SGS* models. The list of *LES* models contains information about models, *LES* filters and filter width functions. In [39], [7] and [40] numerical simulations are carried out on different configurations of *VAWT* by using these methods with two-dimensional and three-dimensional approach. Furthermore in [39] the feasibility and accuracy of different methods based on $2.5D$ simulations is investigated. These simulations consider $3D$ blade of infinite length.
Hybrid RANS-LES method is another approach, which aims to increased physical fidelity over full *RANS* without the costs of a *LES*. The essential idea of hybrid methods is to apply *LES* in the separated flow region and *RANS* in the attached, or weakly separated boundary layers. In recent years, several *hybrid RANS-LES* approaches have been proposed. Spalart *et al.* persecuted the Detached Eddy Simulations (*DES*), in

which *RANS* with the Spalart-Allmaras turbulence model is used.

There are two versions of *DES*: the original model (*DES-SA*) and a variant of a Travin *et al.* based on the *SST* $k-\omega$ turbolence model (*DES-SST*). In *DES*, a hybrid turbolence model that can switch between *RANS* and *LES* modes is obtained by replacing the length scale in the dissipation of either the eddy viscosity (*DES-SA*) or the turbolence kinetic energy (*DES-SST*).

The two other hybrid approaches are Very-Large Eddy Simulation *VLES* of Speziale [8], which is based on a Reynolds-stress transport model, and Scale Adaptive Simulation (*SAS*), which is based on a one-equation turbolence model.

The main issue is the possible dependence on the grid of the boundary between the *RANS* and the *LES* regions. It is difficult to specify the *RANS-LES* boundary a priori for separated flows, where *LES* shoul be active in the separated flow regions and *RANS* in the boundary layers. [9] allows to overlap *RANS* and *LES* regions. The opposite situation, where the *LES* flow becomes a *RANS* flow usually is not problematic as the coherently structured unsteadiness is simply damped and mixed up.

Finally in [7] *DES*-type Extra-*LES* method (*X-LES*) is developed, which is a composite formulation where a single turbolent kinetic energy equation *iSGS* is defined, that "switched" dynamically between the *RANS* and *LES* formulations, depending on the *RANS* lenght scale and the *LES* filter. This formulation is indipendent of the wall distance, which is an ambiguous parameter for *VAWT*.

The hybrid approaches allow to predict separated flows with substantially higher physical fidelity than unsteady *RANS* (*URANS*) and at substantially lower costs than *LES*. In particular *X-LES* has two clear advantages over other hybrid methods: it uses a clearly defined SGS model in *LES* and it does not depend on the wall distance. All the methods listed and described thus far are generically called under-resolved methods to take distance from the direct resolution (*DNS*)of the governing equations.

The accuracy provided by second-order accurate finite volume (*FV*) or finite element schemes, which proved to be very successful for the simulation of a wide variety of applications, is inadequate for many flows of industrial interest: when a long time integration is required, and, more generally, in the simulation of turbulent flows characterized by vortices, eddies and rotating wakes, such as in vertical axis wind turbines.

The limitations of FV methods for turbulent flows simulations described above could be a problem for the $VAWT$ simulation. Higher order accurate methods such as Discontinuous Galerkin (DG) methods have been considered as alternative discretization techniques to overcome these limitations [11], [13], [14], [15], [16]. The solution of the weak or variational form of a problem is approximated by means of piecewise continuous polynomials inside the elements just as in the classical continuous finite element methods (FEM) but with no continuity constraints between neighbouring element. The lack of a global continuity constraint opens the way to the treatment of the solution at element interfaces by the techniques developed in the context of upwind finite volume method, which is in fact a very effective manner to introduce the stabilization required by any FEM method for the solution of purely advective or advection dominated problems. Furthermore, a discontinuous approximation offers greater geometrical flexibility with respect to more traditional globally continuous methods, since high-order accurate schemes can be constructed on arbitrary and possibly non-conforming grids and elements of different order of accuracy can be easily accommodated in the same grid.
The application of the DG space discretization to incompressible fluid flows has been recently considered [17], [18]. In [6], Liu and Shu introduce a DG method for 2D incompressible flows in stream function formulation, where the DG approximation is applied to the momentum equation, while a continuous finite element approximation of the stream function is computed by a standard Poissons solver. Cockburn proposes and thoroughly analyzes the local discontinuous Galerkin (LDG) method for the Stokes, Oseen and incompressible Navier-Stokes equations [12], [19].

In the incompressible $RANS$ equations of $MIGALE$ code the peculiar formulation of the inviscid interface numerical flux, based on the exact solution of the Riemann problem for the artificial compressibility perturbation of the locally $1D$ inviscid Euler equations, provides the necessary coupling between the discretized incompressibility constraint and the rest of the governing equations. Viscous terms are discretized according to the $BR2$

scheme, [46], while the k–ω turbulence model implementation follows the approach proposed in [15] for compressible flows, whereby the use of $\tilde{\omega} = log\omega$ instead of ω guarantees the positivity of ω itself and a locally varying lower bound on $\tilde{\omega}$ in the source terms and in the eddy viscosity constitutive relation enforces the fulfilment of realizability conditions for the turbulent stresses. Furthermore, in order to efficiently address the simulation of flows around bodies rotating at constant angular velocity, such as turbomachinery rotors and propellers, the governing equations can be solved also in a rotating reference frame.

Target

The objective of this project is to develop a 3*D* *DG* solver for incompressible unsteady turbulent flows. In particular, this work will focus on (i) the investigation and implementation of high-order time integration schemes, (ii) the investigation and implementation of preconditioner to enhance the computational efficiency and (iii) the comparison of the results for a *VAWT* by using the *RANS* ($k - \omega$ turbulence model in the high and low Reynolds version) equations, and an *hybrid RANS-LES* approach.
The incompressible turbulent version of the *DG* code *MIGALE* will be extended to 3*D*. The performance of different high-order temporal schemes (Runghe-Kutta-type and Rosenbrock-type) will be investigated. Adaptive time step will be implemented and used to optimize the robustness of the code and the *CPU time*.
The Runge-Kutta scheme allows to use the pseudo-compressibility method in order to accelerate the solution convergence to steady state. When applied to unsteady simulations, this method requires the dual-time stepping technique (*DTS*) to obtain a solution indipendent of the compressibility parameter [4]. Furthermore, different strategies to improve the efficiency of the solver will be evaluated, *i.e.* new preconditioner and the multigrid technique [47] [48].
Finally the *X-LES* method, due to the advantages illustrated in the previous section, will be implemented in the code, assessed for reference computations and finally compared with *RANS* in the computation of a detached flow.

Results

The results of this work will focus on the prediction of the performance and the aerodynamics characteristics of a *VAWT* with a high-order *DG* method. Different numerical simulations will be carried out by using *RANS* equations ($k - \omega$ turbulence model) and the *X-LES* method. The results of the numerical simulations will be compared with the available experimental data.

Advantages of the *X-LES* approach in the simulation of highly detached flow will be demonstrated.

Research plan

The work can be divided into several phases:

1. Literature review for some key topic of the project. *CFD* analysis of *VAWT*, *DG*, incompressible *RANS* equations and $k-\omega$ closure model for turbulent flows, *hybrid RANS-LES* methods, pseudo-compressibility methods, dual-time stepping technique with Runge-Kutta schemes, adaptive time step and preconditioning.

2. Extension of the *DG* code *MIGALE*. The incompressible turbulent version of the *MIGALE* code will be extended to three-dimensional case, by implementing the *RANS* equations with the $k-\omega$ turbulence model for high and low Reynolds number.

3. Evaluation and implementation of different high-order temporal schemes. Different temporal scheme will be compared for different test cases, to investigate their performance in terms of stability and efficiency (*CPU* time). The schemes will be evaluated for the laminar and turbulent flows using two test cases: the travelling waves and the flow around a NACA0018 airfoil.

   In the first test case the laminar solution of travelling dambed waves on a doubly-periodic unit square $[0.25, 1.25] \times [0.5, 1.25]$ is computed. A $\mathbb{P}^6$ solution will be used to keep the space discretization error below the time integration error allowing to numerically assess the design-order accuracy of the time integration schemes. In the second test case the incompressible turbulent flow around a *NACA0018* airfoil is computed. The farfield conditions are $Re = 300000$, $Re_T = 1$, $Tu = 0.1$ and an angle of attack $30°$.

The travelling waves is used to verify the order of convergence of the scheme, where the analytical solution is known. Instead the test case on isolated NACA0018 profile is used to optimize the implementation of the scheme during long-time simulations. Two schemes will be implemented: Rosenbrock-type *ROS3PL* scheme ([34]) and Runge-Kutta-type *ESDIRK46* scheme ([45]).

4. Implementation of pseudo-compressibility method and dual-time stepping technique, for the *ESDIRK* scheme. The effective reduction of the computation time due to these techniques will be evaluated. The same test cases are adopted.

5. Adaptive time step implementation. For unsteady and long-time simulations it is convenient to introduce a strategy to control the time step. This technique allows to optimize the computational time and enhance the robustness of the code. Same test cases adopted for the evaluation of the temporal schemes will be used.

6. Efficiency improvement of the solver. Different strategies will be evaluated to enhance the efficiency: preconditioners and the h-multigrid, as preconditoner and solver. Performance of the proposed strategies will be investigated on the same test cases of point 3.

7. Implementation and investigation of an under-resolved turbulence method. The focus is on *hybrid RANS-LES* methods. In particular the *X-LES* method will be implemented in the code and assessed using the circular cylider test case.

8. Numerical simulation of the *VAWT*. $2D$, $2.5D$ and $3D$ simulations will be performed using the *RANS* and the *X-LES* approaches. The results will be compared with the available experimental data.

Bibliography

[1] Yu-Hong Ran, Jun-Gang Wang *On preconditioned iterative methods for unsteady incompressible Navier–Stokes equations*, Applied Mathematics and Computation 234 (2014), 417-485.

[2] Joachim Rang *Improved traditional Rosenbrock-Wanner methods for stiff ODEs and DAEs*, Journal of Computation and Applied Mathematics 262 (2014) 105-114.

[3] J. Lang, J. Verwer *ROS3PL, An accurate third-order Rosenbrock solver designed for parabolic problems*, BIT (2001), Vol. 41, No.4, 731-738.

[4] Chin Hoe Tai, Yong Zhao *Parallel unsteady incompressible viscous flow computations using an unstructured multigrid method*, Journal of Computational Physics 192 (2003), 277-311.

[5] S. Isono, D. W. Zingg *A Runge-Kutta-Newton-Krylov Algorithm for Fourth-Order Implicit Time Marching Applied to Unsteady Flows.*

[6] C. Liu, X. Zheng, C. H. Sung *Preconditioned multigrid methods for unsteady incompressible flows*, Journal of Computational Physics 139 (1998), 35-57.

[7] J. C. Kok, H. S. Dol, B. Oskam, H. Van der Ven *Extra-Large Eddy simulation of massevely separated flows*, 42nd AIAA Aerospace Sciences Meeting and Exhibit (2004).

[8] C. G. Speziale *Turbolence modelling for time-dependent RANS and VLES: a review*, 36nd AIAA Aerospace Sciences Meeting and Exhibit (1998).

[9] D. Laurence *Large eddy simulation of industrial flows*, Closure strategies for turbolence and transitional flows (2002), 392-406.

[10] *Renewables* 2011*: Global Status Report*, REN21, 2011.

[11] Cockburn, C. W. Shu*The Runge-Kutta local projection P1 discontinuous Galerkin method for scalar conservation laws*, Numer. Anal. 25 (1991), 337-361.

[12] Cockburn B., Kanschat G., Schotzau D., Schwab C *Local discontinuous Galerkin methods for the Stokes system*, SIAM J. Numer. Anal., 40, 319-343.

[13] Bassi F. and Rebay S. *High order accurate discontinuous Galerkin methods in computational fluid dynamics: from model problems to complex turbulent flows-Part I*, 35th CFD/ADIGMA course on very high order discretization methods, 2008.

[14] Bassi F. and Rebay S. *High order accurate discontinuous Galerkin methods in computational fluid dynamics: from model problems to complex turbulent flows-Part II*, 35th CFD/ADIGMA course on very high order discretization methods, 2008.

[15] Bassi F., Botti L., Colombo A., Ghidoni A. and Rebay S.*DG for Turbulent Flows*, DG for Turbulent Flows, 2010.

[16] Bassi F., Crivellini A., Rebay S. and Savini M. *Discontinuous Galerkin solution of the Reynolds-averaged Navier-Stokes and* $k-\omega$ *model equations*, Computer and Fluids 34, 2003.

[17] Bassi F., Crivellini A., Rebay S. and Di Pietro D.A. *An artificial compressibility flux for the discontinuous Galerkin solution of the incompressible Navier-Stokes equations*, Journal of Computational Physics 218, 2006.

[18] Bassi F., Crivellini A. and D'Alessandro V. *An implicit high order discontinuous Galerkin solver for the incompressible RANS and Spalart-Allmaras equations*, Wiley InterScience, 2010.

[19] Cockburn B., Kanschat G., Schotzau D. *The local discontinuous Galerkin method for the Oseen equations*, Math. Comp., 73, 569-593 (2010).

[20] Liu J.G., Shu C.W. *A high-order discontinuous Galerkin method for 2D incompressible flows*, J. Comput. Phys., 160, 577- 596 (2010).

[21] J. Farmer, L. Martinelli, A. Jameson *Fast multigrid method for solving incompressible hydrodynamic problems with free surface*, AIAA J. 32 (1994) 1175-1182.

[22] W. K. Anderson, R. D. Rausch, D. L. Bonhaus *Implicit7multigrid algorithm for incompressible turbulent flows on unstructured grids*, J. Comput Physic 128 (1996) 391-408.

[23] A. Chorin *A numerical method for solving incompressible viscous flow problem*, J. Comput. Physic 2 (1967), 12-26.

[24] C. T. Chan, K. Anastasiou *Solution of incompressible flows with or without a free surface using the finite volume method on unstructured triangular meshes*, Internat J. Numer. Method Fluids 29 (1995) 35-57.

[25] S. E. Rogers, D. Kwak *Upwind differencing scheme for the time accurate incompressible Navier-Stokes equations*, AIAA J. 28 (1990), 253.

[26] Saffman P.G., *A model for Inhomogeneous turbulent flow*, Proc. Roy. Soc., Londra, vol. A317, 1970.

[27] Spalding D.B. e Launder B.E., *Mathematical models for turbulence*, Academic Press, 1972.

[28] Coakley T.J., *Turbulence modeling methods for the compressible Navier-Stokes equations*, AIAA Paper 83-1693, 2003.

[29] Speziale C.G., Abid R. e Anderson E.C., *A critical evaluation of two-equation models for near wall turbulence*, AIAA Paper 90-1481, 1990.

[30] Menter F.R., *Improved two-equation $k-\omega$ turbulence models for aerodynamic flows*, NASA TM-103975, 1992.

[31] Peng S.H., Davidson L. e Holmberg S., *A modified low-Reynolds-number $k - \omega$ model for recirculating flows*, ASME: Journal of Fluids Engineering 119, 1997.

[32] Robinson D.F., Harris J.E e Hassan H.A., *Unified turbulence closure model for axysymmetric and planar free shear flows*, AIAA Journal, vol. 33, no, 12, 1995.

[33] Hinze J.O., *Turbulence*, McGraw-Hill, 1975.

[34] Joachim Rang, *An analysis of the Prothero-Robinson example for constructing new DIRK and ROW methods*, Journal of Computational and Applied Mathematics 262 (2014), 105-114.

[35] Mark H. Carpenter, David Gottlieb, Saul Abarbanel, Wai-Sun Don, *The teorica accuracy of Runge-Kutta time discretizations for the initial boundary value problem: a careful study or the boundary error*, NASA contractor Report 191561 (1993).

[36] D. Pathria *The Correct Formulation of Intermediate Boundary Conditions for Runge-Kutta Time Integration of Initial Boundary Value Problems*, SIAM J. Sci. Comput., 18(5), 1255–1266.

[37] S. Beck, R. Weiner, H. Podhaisky, B.A. Schmitt. *Implicit peer methods for large stiff ODE systems.*, J. Appl. Math. Comp. 38, 389-406, (2012).

[38] Sigal Gottlieb *On high order strong stability preserving Runge-Kutta and Multistep time discretizations*, Journal of scientific Computing, 25 (2005).

[39] Chao Li, Songye Zhu, Tou-Lin Xu *2,5D Large eddy simulation of vertical axis wind turbine in high angle of attack flow.*

[40] L. X. Zhang, Y. B. Liang, X. H. Liu *Aerodynamics performance prediction of straight-bladed vertical axis wind turbine based on CFD*, Advances in Mechanical Engineering (2013).

[41] Mohamed S. Ebeida *Unsteady incompressible flow simulating using Galerkin finite elements with spatial/temporal adaptation*, American Institute of Aeronautics and Astronautics.

[42] P. O. Persson, J. Peraire *Newton-GMRES preconditoning for discontinuous Galerkin discretizations of the Navier-Stokes equations.*

[43] Laslo T. Diosady *A linear multigrid preconditioner for the solution of the Navier-Stokes equations using a discontinuous galerkin discretization*, Master of science thesis in computational for design and optimization.

[44] Cristian R. Nastase, Dimitri J. Mavriplis *High order discontinuous Galerkin methods using an hp-multigrid approach*, Journal of computational physics 213 (2006), 330-357.

[45] Kanevsky A., Carpenter M. H., Gottlieb D., Hesthaven J. S. *Application of implicit-explicit high order Runge-Kutta methods to discontinuous Galerkin schemes*, Journal of computational physics 225 (2007), 1753-1781.

[46] F. Bassi, S. Rebay, G. Mariotti, S. Pedinotti, M. Savini *A high-order accurate discontinuous finite element method for inviscid and viscous turbomachinery flows*, Proceedings of the 2nd European Conference on Turbomachinery Fluid Dynamics and Thermodynamics, Technologisch Instituut, Antwerpen, Belgium, 1997, pp. 99–108.

[47] A. Belov, L. martinelli and A. Jameson *A new implicit algorithm with multigrid for unsteady incompressible flow calculations*, 33rd Aerospace Sciences Metting and Exhibit, AIAA 95-0049.

[48] J. Farmer, L. martinelli and A. Jameson *A fast multigrid method for solving incompressible hydrodynamic problems with free surfaces*, 31st Aerospace Sciences Metting and Exhibit, AIAA 93-0767.