

Extended summary

Numerical Solutions of Turbulent Flows: industrial applications

Curriculum: Energy Sciences

Author

Valerio D'Alessandro

Tutor(s)

Advisor: Prof. Ing. Renato Ricci Coadvisor: Dott. Ing. Andrea Crivellini

Date: 8-February-2013

Abstract. The study of innovative energy systems often involves complex fluid flows problems and the Computational Fluid Dynamics (CFD) is one of the main tools of analysis.

It is very easy to understand as developing new high-accuracy solution techniques for the fluid flow governing equations is of an extreme interesting research area.

This work is aimed in the field of numerical solution of turbulent flows problems in industrial configurations with standard and innovative discretization techniques. In this thesis great efforts were addressed in to develop of a high-order Discontinuous Galerkin (DG) solver for incompressible flows in order to enjoy its accuracy in a wide class of industrial problems. DG methods are based on polynomial approximations inside the computational elements with no global continuity requirement and they are receiving an increasing interest in CFD community because they combine flexibility, stability, robustness and accuracy features. Starting from a 2D viscous version of a code, based on the artificial compressibility flux DG method [1], in this thesis a 3D version is presented and its suitability for DNS computations is demonstrated. Moreover the Spalart-Allmaras (SA) turbulence model has been implemented in both the 2D and 3D solvers. It is worth noting that DG space discretization of RANS equations is a difficult task due the numerical stiffness of the equations. In this work the SA model is modified in source and diffusion terms to deal with numerical instabilities coming-up when the working variable, or one of the model closure functions, become negative thus unphysical. It is important to remark that in the present literature are not reported others DG solvers for the incompressible RANS-SA system.



The realiability, accuracy and robustness of the solution method was assessed computing several test-cases in simple and real-life configurations. Simultaneously unsteady Aerodynamics of the Savonius wind rotor and the flow field inside a Ranque-Hilsch vortex tube (RHVT) were extensively studied with standard finite volume solvers obtaining innovative results.

The flow problem around a Savonius wind rotor is a two-way coupling type, i.e. the reciprocal influences between solid body and the fluid must be considered.

The strategy used to solve this issue uses a SMM (Sliding Mesh Model) approach for the fluid while the solid body motion is treated solving the rotor dynamics by means of a custom MatLab algorithm.

RHVT is a simple device in which a compressed gas flow is split into two low pressure flows whose temperatures are respectively higher and lower than the one of the inlet flow. This effect, called the Ranque-Hilsch effect or "thermal separation", is merely a fluid dynamic process as it takes place in a device with no moving parts and it is not still completely understood. Its computation is a challenging task hence an accurate numerical simulation of the flow field inside a RHVT is here conducted.

Nevertheless in this moment our DG solvers can cover a wide range of Reynolds numbers, they have not still found application to analyze problems as Savonius rotors or RHVT since at the time of those analysis our codes cannot deal with that kind of flows.

Keywords. Computational Fluid Dynamics, Discontinuous Galerkin method, Ranque-Hilsch vortex tube, Savonius wind rotor, Spalart-Allmaras turbulence model.

1 Introduction

This work is aimed in the field of numerical solution of turbulent flows problems in industrial configurations; both standard and innovative solution techniques were used to face the problems here stated.

In the thesis great efforts were addressed to the develop of a high-order Discontinuous Galerkin (DG) solver for incompressible flows in order to enjoy of its accuracy in a wide class of industrial problems. In recent years DG methods for Computational Fluid Dynamics (CFD) received an increasing interest because they combine flexibility, stability, robustness and accuracy features. DG methods are based on polynomial approximations inside the computational elements with no global continuity requirement. As in continuous finite elements methods, DG methods can increase the order the accuracy of the solution raising the degree of polynomial approximation while its accuracy is not affected by the use of unstructured hybrid meshes useful for complex configurations. The discontinuous approximation between neighboring elements is treated using an upwind schemes for interface fluxes as in finite volume methods. The resulting compactness of the algorithm makes the method easily and efficiently implementable on parallel architectures. Despite these attractive advantages the suitability of DG methods for analyze complex engineering problems remains a challenging tasks.

Starting by a 2D viscous version of a code, based on the artificial compressibility DG method [1], in this thesis a 3D version of the same code is presented. This solver was also equipped with statistical tools in the perspective of using the code in Direct Numerical Simulations (DNS) field; a preliminary DNS of the flow past a sphere at Re = 1000 is presented (these results for brevity are not reported here). Hence the Spalart-Allmaras (SA) turbulence model was implemented in both the 2D and 3D codes. It worth to note that DG space discretization of RANS equations is a difficult task but the very high computational cost related to DNS and Large-Eddy Simulations (LES) makes Reynolds-Average Navier-Stokes (RANS) approach the only feasible for many flow problems. DG solution of RANS equations faces severe problems related to the numerical stiffness induced by the highly non-linear source terms of the turbulence model equations and by grid stretching needed to resolve the near-wall behavior of the turbulent quantities. Thus, high-order DG methods for the RANS and turbulence model equations must be manipulated with some form of limiting or stabilization terms in order to prevent blow-up of computations. In this work a novel implementation of the SA model was introduced, and published by Author (in collaboration with Crivellini and Bassi) in [2]. It deals with negative $\tilde{\nu}$ values by modifying the source and diffusion terms in the SA model equation only when the working variable or one of the model closure functions become negative thus unphysical. This results in an efficient high-order implementation where either stabilization terms or even additional equations are avoided. The realiability, accuracy and robustness of the solution method was assessed computing several test-cases in simple and complex configurations.

Simultaneously unsteady Aerodynamics of the Savonius wind rotor the flow field inside a Ranque-Hilsch vortex tube (RHVT) were extensively studied with Finite Volume solvers. More in depth unsteady Aerodynamics of the Savonius wind rotor was studied solving RANS equations with a commercial CFD solver. The flow problem around a Savonius wind rotor is a two-way coupling type, i.e. reciprocal influences between solid body and the fluid must be considered. The strategy used to solve this issue uses a SMM (Sliding Mesh



Model) approach for the fluid while the solid body motion is treated solving the rotor dynamics by means of a custom MatLab algorithm.

The numerical results are in very good agreement with experimental measurements performed at Environmental Wind Tunnel of the Università Politecnica delle Marche.

In the following the complex flow field developing inside a commercial Ranque-Hilsch vortex tube (RHVT) was extensively studied. RHVT is a simple device in which a compressed gas flow is split into two low pressure flows whose temperatures are respectively higher and lower than the one of the inlet flow. This effect, called the Ranque-Hilsch effect or "thermal separation", is merely a fluid dynamic process as it takes place in a device with no moving parts. This interesting phenomenon is not completely understood and its computation is a challenging task. Many efforts to explain the thermal separation phenomenon have been made in the past, based on theoretical, numerical and experimental analysis. Nevertheless this phenomenon is not completely understood and its computation is a challenging task. On the grounds of the above mentioned observations stems the present research activity whose aim is the development of an accurate numerical simulation of the flow field and its physical characteristics inside a RHVT. Both RANS and LESs simulations were carried out.

Even if in this moment our DG solver can cover a wide range of Reynolds numbers it was not here adopted to analyze industrial problems, such as Savonius wind rotors and/or Ranque-Hilsch Vortex Tubes. In fact at the time of those analysis our DG solver could not deal with that kind of flow fields since the develop here reported was just at its beginning. Future work will be devoted to the application of our solver to these topics, for the part of industrial applications, while the extension of the DG method to and to Detached Eddy Simulations (DES) and LES will be the main subject of DG research area.

All the solutions have been obtained on distributed memory parallel machines: the computations with lower computational requests were run on a Linux Cluster, with 8 AMD Opteron based nodes for a total of 64 CPU cores operating at 2.4 GHz, while the more expensive ones were performed on the IBM SP6, BlueGene/P or BlueGene/Q-FERMI supercomputing facilities at CINECA using up to 2048 CPU cores.

2 DG Solution of the Spalart-Allmaras model

Discontinuous Galerkin (DG) methods for Computational Fluid Dynamics have been first applied to strictly hyperbolic problems as the Euler equations [3, 4]. After the pioneering work of Bassi and Rebay [5], DG methods have received much attention also for the high-order discretization of elliptic problems and, since then, many stable DG discretization schemes for the diffusive terms of the compressible Navier-Stokes equations have been developed. More recently, DG methods suited for the incompressible Navier-Stokes (INS) equations have been proposed and analyzed in a number of papers [6, 7, 8, 9, 10, 11, 12, 13, 14].

Papers about the high-order DG space discretization of the Reynolds-Averaged Navier-Stokes (RANS) equations are far less numerous and they only deal with compressible flows. This is to be related to severe problems of numerical stiffness induced by the highly non-linear source terms of the turbulence model equations and by grid stretching needed to resolve the near-wall behavior of the turbulent quantities. Thus, high-order DG methods for the RANS and turbulence model equations must be carefully set up and always include some form of limiting or stabilization terms in the turbulence model equations in order to prevent blow-up of computations.



In this work a high-order DG method for the incompressible RANS equations coupled with the SA turbulence model (1) has been presented.

$$\nabla \cdot \mathbf{u} = 0$$

$$\frac{\partial \mathbf{u}}{\partial t} + \nabla \cdot (\mathbf{u} \otimes \mathbf{u}) + \nabla p = \nabla \cdot ((\nu + \nu_T) (\nabla \mathbf{u} + \nabla \mathbf{u}^T)) = \mathbf{0}$$

$$\frac{\partial \widetilde{\nu}}{\partial t} + \nabla \cdot (\mathbf{u} \widetilde{\nu}) - \frac{1}{\sigma} \nabla \cdot (\eta \nabla \widetilde{\nu}) = s$$
(1)

where $p = P/\rho$ is pressure dived by the density and the turbulent viscosity, v_t , is computed, according to the \tilde{v} variable as:

$$v_T = f_{v1} \widetilde{v}$$
.

In the above equation the source term, *s*, is given by:

$$s = c_{b1}\widetilde{S}\widetilde{v} + \frac{c_{b2}}{\sigma}\nabla\widetilde{v} \cdot \nabla\widetilde{v} - c_{w1}f_{w1}\left(\frac{\widetilde{v}}{d}\right)^2$$

where *d* is the minimum distance from the wall and \tilde{S} , which is a function of both the vorticity magnitude *S* and the turbulent variable \tilde{v} is the following production term:

$$\widetilde{S} = S + \frac{\widetilde{v}}{k^2 d^2} f_{v_2} \qquad \qquad \mathbf{\Omega} = \frac{\nabla \mathbf{u} - \nabla \mathbf{u}^T}{2} \qquad \qquad S = \sqrt{2\mathbf{\Omega} : \mathbf{\Omega}}$$

To completely define the PDEs the closure functions:

$$\chi = \frac{\tilde{v}}{v} \qquad f_{v1} = \frac{\chi^3}{\chi^3 + c_{v1}^3} \\ f_{v2} = 1 - \frac{\chi}{1 + \chi \cdot f_{v1}} \qquad f_w = g \left(\frac{1 + c_{w3}^6}{g^6 + c_{w3}^6}\right)^{\frac{1}{6}} \\ g = r + c_{w2} \left(r^6 - r\right) \qquad r = \frac{\tilde{v}}{\tilde{S}k^2 d^2}$$

and closure constants:

$$c_{b1} = 0.1355 \qquad c_{b2} = 0.622 \qquad c_{v1} = 7.1$$

$$\sigma = 2/3 \qquad c_{w1} = \frac{c_{b1}}{k^2} + \frac{(1 + c_{b2})}{\sigma} \qquad k = 0.41$$



are required. The diffusion coefficient of the SA model equation reads:

 $\eta = \nu + \widetilde{\nu}$.

The resulting turbulence model is the standard SA model without the trip function, hence employed in a fully turbulent mode.

The occurrence of negative eddy viscosity values has been addressed and numerically evaluated by means two approaches. The first one (2), (3) was suggested by Allmaras himself

$$s = \begin{cases} c_{b1} \widetilde{S} \widetilde{v} + \frac{c_{b2}}{\sigma} \nabla \widetilde{v} \cdot \nabla \widetilde{v} - c_{w1} f_{w1} \left(\frac{\widetilde{v}}{d}\right)^2 & \chi \ge 0 \\ c_{b1} S \widetilde{v} g_n + \frac{c_{b2}}{\sigma} \nabla \widetilde{v} \cdot \nabla \widetilde{v} - c_{w1} \left(\frac{\widetilde{v}}{d}\right)^2 & \chi < 0 \end{cases}$$
(2)

$$\eta = \begin{cases} \nu(1+\chi) & \chi \ge 0 \\ \nu\left(1+\chi+\frac{1}{2}\chi^2\right) & \chi < 0 \end{cases}$$
(3)

$$s = \begin{cases} c_{b1} \widetilde{S} \widetilde{v} + \frac{c_{b2}}{\sigma} \nabla \widetilde{v} \cdot \nabla \widetilde{v} - c_{w1} f_{w1} \left(\frac{\widetilde{v}}{d}\right)^2 & \chi \ge 0\\ 0 & \chi < 0 \end{cases}$$
(4)

$$\eta = \begin{cases} \nu (1 + \chi) & \chi \ge 0 \\ \nu & \chi < 0 \end{cases}$$
(5)

while the second one (4), (5) was inspired to the DG implementation of the k- ω turbulence model reported in Bassi et al. [15]. The two approaches differ in that the corrections proposed by Allmaras are continuous and ensure continuity of the Jacobian, while in the approach of Bassi et al. [15] the Jacobian is discontinuous. We remark that, since the Allmaras modifications include more terms that could lead to the growth of an unphysical $\tilde{\nu}$ value, usually the zones with negative eddy viscosities are smaller in size. However the overall quality of results and the behavior of the solution algorithm are quite similar in the two approaches.

As a second contribution to a robust and efficient implementation of the SA model, we have introduced a new limiting procedure for the closure function r of the model, (6). The behavior of this function in the original SA model sometimes leads to a positive Jacobian



of the source term which impairs the convergence to steady state of an implicit solver. The procedure introduced in this work prevents the r function, whose square root is in fact a length scale, from attaining negative values, and avoids the sign inversion of production and destruction source terms.

$$r^{*} = \frac{\widetilde{v}}{\widetilde{S}k^{2}d^{2}} \qquad r = \begin{cases} r_{\max} & r^{*} < 0 \\ r^{*} & 0 \le r^{*} < r_{\max} \\ r_{\max} & r^{*} > r_{\max} \end{cases}$$
(6)

Numerical experiments have shown that this modification of the SA model becomes essential when computing flows with strong adverse pressure gradients or large recirculation and separation zones.

By comparing our results with reference ones available in the literature we have found that all the SA model modifications here proposed do not affect the RANS-SA solution, while they significantly improve the reliability of the model and the robustness of the code. Furthermore, these SA modifications are not specific to the DG method, thus they can be easily implemented in any RANS-SA solver with significant advantage. Finally, the proposed modifications do not require any additional computational effort and thus do not impair the efficiency of the implicit solver, which is able to provide high-order solutions in a small number of Newton steps.

3 Results

3.1 Discontinuous Galerkin computations

DG solution of the 3D incompressible Navier-Stokes and the RANS-SA system was achived using the artificial compressibility DG flux method [1]. The main feature of the method is an original formulation of the inviscid interface numerical fluxes based on the solution of Riemann problems with a relaxed incompressibility constraint which allows to recovery the hyperbolic nature of the equations. The viscous numerical fluxes are handled with the well-established BR2 scheme.

Differential Algebraic Equations system resulting by DG space discretization is solved by means of linearly implicit (Rosenbrock-type) Runge-Kutta scheme.

To solve the linear system we resort to both the matrix based or matrix-free [16] preconditioned GMRES (Generalized Minimal RESidual) linear solvers available in PETSc library [17], the software upon which our DG code relies for the purpose of parallelization. The SPMD (single process multiple data) strategy is based on grid partitioning accomplished by means of the METIS package, [18]. Each processor owns the data related to its local portion of the grid and the data on remote processors are accessed through MPI paradigm. Thanks to the compactness of our DG method the data owned only by the near neighbor elements at partition boundaries need to be shared among different processes.

The reliability, robustness and accuracy of the proposed implementation have been assessed by computing several high Reynolds number turbulent 2D test cases: the flow over a flat plate ($Re=10^7$), the flow past a backward-facing step (Re=37400) and the flow around a



NACA 0012 airfoil (Figure 1) at different angles of attack ($a=0^{\circ},10^{\circ},15^{\circ}$) and Reynolds numbers (Re=2.88 $\cdot 10^{\circ},6\cdot 10^{\circ}$).



Velocity Magnitude contour plot Pressure coefficient Figure1: NACA0012 Re=2.88 · 10⁶.

The Spalart-Allmaras turbulence model implementation, here presented, was also to predict the Laminar Seprations Bubble (LSB) on low Reynolds numbers operating airfoils. In particular the Selig-Donovan (SD) 7003, E387, WT1 and WT2 airfoils were investigated. WT1 and WT2 airfoils were developed by the Thermal Fluid-Dynamics section of the Università Politecnica delle Marche. This the subject of a forthcoming paper.

Several high-order DG computations of steady problems in 3D simple and in real life configurations were also performed: flow over a 3D sinusoidal bump in a channel (Re= $3\cdot10^6$), flow past a sphere (Re= $1.14\cdot10^6$), see Figure 2 and 3, and the flow around the DLR-F6 wing body transport configuration (Re= 10^6 , $\alpha=1^\circ$). Lastly the flow past a delta wing was extensively computed (Figure 4 and 5). The case defined by Chu and Luckring in [19], in a configuration named VFE-2 investigated within the framework of an RTO Task Group, that corresponds to the geometry of a 65° sweep angle and a blunt leading edge. The results concerning this cases were published by the Author in [20,21]



Pressure coefficient contour plot $\tilde{\nu}$ contour plot Figure 2: Sphere Re=1.14·10⁶. P6 solution.





Skin friction coefficient Pressure coefficient Figure 3: Sphere Re=1.14·10⁶.



Figure 4: Pressure distribution at Re=10⁶. Lines: numerical P₅ results on the; circles: experimental data





Figure 5: Re=10⁶, $a=18^{\circ}$. Streamlines: contours plot of pressure at the wall and of eddy viscosity in the wake region.

The 3D viscous version of the DG code was also extensively tested in [22, 23] by the Author computing the flow field past a sphere at moderate Reynolds numbers. We have also proved that the solver can be used a DNS solver for turbulent flows. The results are not reported here for the sake of brevity.

3.2 Unsteady Aerodynamics of a Savonius wind rotor

The Savonius wind turbine is a vertical axis wind turbine (VAWT) created for the first time by the finnish engineer Sigurd Savonius in 1925. Aerodynamic theories developed in order to analyze horizontal axis wind turbines, as the BEM (Blade Element Momentum) theory [24] are able to predict the performances of VAWT driven by lift (i.e. Darrieus rotors) only. Owing to the different nature of the flow field around the Savonius rotor, the above mentioned methods cannot be applied in its analysis.



Figure 6: Contour of turbulence intensity (Experimental value: $\lambda = 0.735$)



For these reasons wind tunnel tests and computational techniques are the only tools available for studying Savonius wind rotors.

The strategy used in this work to solve the flow problem around the Savonius rotor uses a SMM (Sliding Mesh Model) approach for the fluid while the solid body motion is treated solving the second cardinal equation of dynamics by means of a custom MatLab numerical algorithm able to import CFD data, to calculate the rotor angular velocity and to provide this variable as input to the CFD code.

Time marching of the solution of the second cardinal equation has been executed using an Euler method in the initial steps and a four stage explicit Runge-Kutta (or an Adams-Bashfort) scheme in the following steps.

The experimental data were used to validate the developed computational methodology. The comparison of performance data obtained by numerical simulations and experimental measurements show a very good agreement, Figure 7.

This suggest to use this numerical method for studying new blades shapes in order to produce an increasing of the rotor performance. Moreover the results exhibit by numerical simulations allowed to gain an insight of the flow field mean features.



Power coefficient Tip-speed ratio Fig 7: Comparison between numerical and experimental rotor performance

3.3 Fluid Dynamic Analysis of a Ranque-Hilsch Vortex Tube

The Ranque-Hilsch vortex tube (RHVT) is a simple device in which a compressed gas flow is split into two low pressure flows whose temperatures are respectively higher and lower than the one of the inlet flow. This effect, called the Ranque-Hilsch effect or "thermal separation", is merely a fluid dynamic process as it takes place in a device with no moving parts.

The RHVT consists of a circular tube with an inlet orifice, where compressed gas flows tangentially through several nozzles, azimuthally arranged. The high pressure flow affected from very strong swirling motion, is split into two streams of different temperatures: the hotter one spiraling in touch with the wall in the outer zone of the tube, the colder one spiraling in the opposite direction close around the central axis, Figure 8.

The hot and cold gas streams leave the device through two axial outlet orifices that can be arranged either both on the same side of the tube, (uniflow vortex tube), or on the opposite



sides of the tube, (counter-flow vortex tube). The mass flow rate is regulated by a coneshaped control valve, placed near the hot exit. This valve can vary the mass flow rate leaving the hot exit influencing the temperature of the gas leaving the device.

The CFD simulation of the flow field in a Ranque–Hilsch vortex tube is a challenging task because of its compressibility, turbulence and high swirl.



Figure 8: Components of commercial Vortex Tube: 1-air inlet;2-hot exit; 3-control valve; 4-main body; 5-vortex chamber; 6-generator; 7-cold exit; 8- threaded ring nut; 9-brass inset

In the present work an accurate numerical simulation of the flow field and its physical characteristics inside a commercial RHVT was performed using the CFD solver FLU-ENT. Several different approaches to the simulation of the turbulence (RANS and LES) were tested. Furthermore, different turbulence models were used as RANS equations closures: RNG $k - \varepsilon$ model and a linear RSM. Large Eddy Simulations of the internal flow, using the Smagorinsky's sub-grid model, were performed too.

RANS simulations were performed on an axisymmetric computational domain, while LES were performed on a (complete) three-dimensional computational model in order to avoid symmetry imposing in turbulent structures.

Results showed that flow in the tube is split into two helical co-axial streams, with different thermal features, placed near the internal wall of the tube, the hot one, and near the axis, the cold one.

Flow patterns and velocity profiles (Figure 9 and 10) in different sections of the tube show a qualitative good agreement with the results available by previous works. Strong differences between results obtained by RNG $k - \varepsilon$ and RSM models are showed in the axial velocity profiles, far from the hot outlet and in the swirl velocity profiles, far from the inlet. This result could be expected as turbulence models features. Simulation with RSM model predicts an axial velocity profile, close to a fully developed channel flow one, while LES shows a "camel's hump" trend for axial velocity profile. LES and RANS simulations, in the version with RSM model, are able to predict a velocity profile similar to a "Rankine Vortex" one for the swirl velocity component. All the simulations confirmed that radial velocity values are very small when compared with axial and tangential ones; hence, this component could be ignored in the analysis of the thermal separation process as in a recent numerical work [25]. RANS simulations proved capable of predicting secondary circulation flow inside a RHVT although this approach showed a single re-circulating vortex structure that extends all over along the tube length. LES can simulate secondary circulation flow



too; anyway a more complex secondary vortex structure appears in this case, due to the three-dimensional and unsteady features of the flow field.



Figure 9: Swirl velocity profiles in RANS and LES simulations at different sections



Figure 10: Axial velocity profiles in RANS and LES simulations at different sections



4 Conclusions

In this work the high-order artificial compressibility flux DG method for the incompressible RANS equations coupled with the SA turbulence model has been presented.

The DG discretization of the nonlinear convective terms is based on the artificial compressibility flux formulation introduced few years ago for the incompressible Navier-Stokes equations [1]. This approach has proved to be very well suited to deal with the numerical stiffness induced by highly stretched grids, with possibly curved elements, and by the strongly nonlinear character of the governing equations, typical of high- Reynolds number turbulent flow computations.

DG implementation and solution of the RANS-SA system were presented here and two are the main contributions to this topic. The first one is how to deal with unphysical, but likely to occur numerically, negative eddy viscosity values. The second one is the proposal, based on a detailed analysis, of a new limiting procedure for one of the model closure functions that strongly affects the numerical stability of the implementation, even for positive values of $\tilde{\nu}$.

By comparing the results here reported with reference results available in the literature we have found that all the SA model modifications do not affect the RANS-SA solution, while they significantly improve the reliability of the model and the robustness of the code.

Furthermore, these SA modifications are not specific to the DG method, thus they can be easily implemented in any RANS-SA solver with significant advantage. Finally, the proposed modifications do not require any additional computational effort and thus do not impair the efficiency of the implicit solver, which is able to provide high-order solutions in a small number of Newton steps.

The code for three-dimensional flows in complex geometries has also demonstrated robustness similar to the two-dimensional case.

Moreover in this work a version of the code for three-dimensional viscous problems was also presented but, for brevity, the results are not reported in this paper.

Simultaneously to the previous activities unsteady Aerodynamics of the Savonius wind rotor and the flow field inside a Ranque-Hilsch vortex tube (RHVT) were extensively studied with commercial Finite Volume solvers. The CFD simulation of the flow field in a Ranque-Hilsch vortex tube is a challenging task because of its compressibility, turbulence and high swirl. Hence, several different approaches to the simulation of the turbulence (RANS and LES) were tested. Furthermore, different turbulence models were used as RANS equations closures: RNG $k - \varepsilon$ model and a linear RSM. Large Eddy Simulations of the internal flow, using the Smagorinsky's sub-gridmodel, were performed too. RANS simulations were performed on an axisymmetriccomputational domain, while LES were performed on a (complete) three-dimensional computational model in order to avoid symmetry imposing in turbulent structures.

Results showed that flow in the tube is split into two helical co-axial streams, with different thermal features, placed near the internal wall of the tube, the hot one, and near the axis, the cold one.

Flow patterns and velocity profiles in different sections of the tube show a qualitative good agreement with the results available by previous works. Strong differences between results obtained by RNG $k - \varepsilon$ and RSM models are showed in the axial velocity profiles, far from the hot outlet and in the swirl velocity profiles, far from the inlet. This result could be expected as turbulence models features. Simulation with RSM model predicts an axial velocity profile, close to a fully developed channel flow one, while LES shows a "camel's hump" trend for axial velocity profile. LES and RANS simulations, in the version with



RSM model, are able to predict a velocity profile similar to a "Rankine Vortex" one for the swirl velocity component. All the simulations confirmed that radial velocity values are very small when compared with axial and tangential ones; hence, this component could be ignored in the analysis of the thermal separation process as in a recent numerical work [21]. RANS simulations proved capable of predicting secondary circulation flow inside a RHVT although this approach showed a single re-circulating vortex structure that extends all over along the tube length. LES can simulate secondary circulation flow too; anyway a more complex secondary vortex structure appears in this case, due to the three-dimensional and unsteady features of the flow field.

In the prediction of the temperature field, RANS models show radial temperature profiles very close between them far from the inlet, while LES predicts a lower temperature near the tube axis and a considerably different static temperature radial profile.

Temperature radial profiles demonstrate a qualitative good agreement with data by previous works. The influence of the computational model used, was evaluated comparing numerical results obtained by RANS simulation in the case of axial and radial hot outlet. Simulations demonstrate that RSM previsions are unaffected by the computational hot outlet variation, while in RNG results some effects can be underlined in axial velocity and temperature profiles, particularly near the inlet. A computational approach able to calculate the flow field around a Savonius wind rotor was also developed. RANS equations were solved in order to obtain information about the flow field. The rotor blades were treated as rigid bodies and their behaviour were modelled by means of the Second Cardinal Equation of Dynamics. An extended wind tunnel testing program was also conducted in the Environmental Wind Tunnel of the Università Politecnica delle

Marche on a split-type Savonius wind rotor. The experimental data were used to validate the developed computational methodology. The comparison of performance data obtained by numerical simulations and experimental measurements shows very goodagreement. This suggests the use of this numerical method for studying new blades shapes in order to produce better rotor performance.

It is important to remark that in this thesis great efforts were addressed to the developing of high-order Discontinuous Galerkin (DG) solvers for incompressible flows in order to enjoy its accuracy in a wide class of industrial problems. Nevertheless in this moment our DG solvers can cover a wide range of Reynolds numbers but they have not still found application to analyze industrial problems, such as Savonius wind rotors and/or Ranque-Hilsch Vortex Tubes, since at the time of those analysis our codes cannot deal with that kind of flow fields. Moreover our modelling technology should be increased in order to correctly analyze rotating flow fields, introducing for example some features such as sliding mesh. Future work will be devoted to develop this aspects and to Detached Eddy Simulationsi, based on the Spalart-Allmaras model, and to LES fields.



References

- F. Bassi, A. Crivellini, D.A. Di Pietro and S. Rebay. An artificial compressibility flux for the disconinuos Galerkin solution of the incompressible Navier-Stokes equations. J. Comput. Phys., 218 (2): 794-815, 2006.
- [2] A. Crivellini, V. D'Alessandro, F. Bassi. A Spalart-Allmaras turbulence model implementation in a discontinuous Galerkin solver for incompressible flows. J. Comput. Phys. . In Press.
- [3] B. Cockburn and C.W. Shu. The Runge-Kutta discontinuous Galerkin finite element method for conservation laws V: Multidimensional systems. J. Comput. Phys., 141:199–224, 1998.ù
- [4] F. Bassi and S. Rebay. High-order accurate discontinuous finite element solution of the 2D Euler equations. J. Comput. Phys., 138:251–285, 1997.
- [5] F. Bassi and S. Rebay. A high-order accurate discontinuous finite element method for the numerical solution of the compressible Navier-Stokes equations. J. Comput. Phys., 131:267–279, 1997.
- [6] J.G. Liu and C.W. Shu. A high-order discontinuous Galerkin method for 2D incompressible flows. J. Comput. Phys., 160(2):577–596, 2000.
- [7] B. Cockburn, G. Kanschat, D. Schötzau, and C. Schwab. Local discontinuous Galerkin methods for the Stokes system. SIAM J. Numer. Anal., 40(1):319–343, 2002.
- [8] B. Cockburn, G. Kanschat, and D. Schötzau. The local discontinuous Galerkin method for linearized incompressible fluid flow: a review. Comput. & Fluids, 34:491–506, 2005.
- [9] B. Cockburn, G. Kanschat, and D. Schötzau. A locally conservative LDG method for the incompressible Navier-Stokes equations. Math. Comp., 74:1067–1095, 2005.
- [10] P.F. Fischer K. Shahbazi and C.R. Ethier. A high-order discontinuous Galerkin method for the unsteady incompressible Navier-Stokes equations. J. Comput. Phys., 222(1):391–407, 2007.
- [11] N.C. Nguyen, J. Peraire, and B. Cockburn. An implicit high-order hybridizable discontinuous Galerkin method for the incompressible Navier-Stokes equations. J. Comput. Phys., 230(4):1147–1170, 2011.
- [12] L. Botti and D.A. Di Pietro. A pressure-correction scheme for convection dominated incompressible flows with discontinuous velocity and continuous pressure. J. Comput. Phys., 230(3):572–585, 2011.
- [13] R. Henniger, D. Obrist, and L. Kleiser. High-order accurate solution of the incompressible Navier-Stokes equations on massively parallel computers. J. Comput. Phys., 229(10):3543– 3572, 2010.
- [14] E. Ferrer and R.H.J. Willden. A high order Discontinuous Galerkin-Fourier incompressible 3D Navier-Stokes solver with rotating sliding meshes. J. Comput. Phys., 231(21):7037 – 7056, 2012.
- [15] F. Bassi, A. Crivellini, S. Rebay, and M. Savini. Discontinuous Galerkin solution of the Reynolds averaged Navier-Stokes and k ω turbulence model equations. Computers & Fluids, (34):507–540, 2005.
- [16] A. Crivellini and F. Bassi. An implicit matrix-free discontinuous Galerkin solver for viscous and turbulent aerodynamic simulations. Computers & Fluids, 50(1):81 93, 2011.
- [17] S. Balay, K. Buschelman, W. D. Gropp, D. Kaushik, M. G. Knepley, L. C. McInnes, B. F. Smith, and H. Zhang. PETSc Web page, 2001. http://www.mcs.anl.gov/petsc.
- [18] G. Karypis and V. Kumar. METIS, a software package for partitioning unstructured graphs, partitioning meshes, and computing fill-reducing orderings of sparse matrices. Technical Report Version 4.0, University of Minnesota, Department of Computer Science/Army HPC Research Center, 1998.
- [19] J. Chu and J.M. Luckring. Experimental surface pressure data obtained on 65° delta wing across Reynolds and Mach number ranges. Technical Report 4645, NASA, February 1996.



Doctoral School on Engineering Sciences

- [20] A. Crivellini, V. D'Alessandro, F.Bassi. High-order discontinuous Galerkin solutions of threedimensional incompressible RANS equations. Computers and Fluids. Accepted.
- [21] A. Crivellini, V. D'Alessandro, F. Bassi. High-order discontinuous Galerkin RANS soultions of the flow field past a delta wing. Computers and Fluids. Submitted.
- [22] A. Crivellini, V. D'Alessandro, F. Bassi, R. Ricci. High-order discontinuous Galerkin computations of the ow past a sphere up to Re=500 The 9th European Fluid Mechanics Conference, 9-13 September 2012, University of Rome "Tor Vergata".
- [23] A. Crivellini, V. D'Alessandro, F.Bassi. Assessment of a high-order discontinuous discontinuous Galerkin solutions of Galerkin method for three-dimensional Navier-Stokes equations: benchmark results for the flow past a sphere up to Re=500. Computers and Fluids. Submitted.
- [24] M.O.L. Hansen. Aerodynamics of Wind Turbines. Earthscan, London, VA, 2008.
- [25] U. Behera, P.J. Paul, K. Dinesh, and S. Jacob. Numerical investigations on flow behaviour and energy separation in Ranque–Hilsch vortex tube. Int.l J. of Heat and Mass Transfer, 51(25– 26):6077 – 6089, 2008.

