11th World Congress on Computational Mechanics (WCCM XI)
5th European Conference on Computational Mechanics (ECCM V)
6th European Conference on Computational Fluid Dynamics (ECFD VI)
July 20 - 25, 2014, Barcelona, Spain

APPLICATION OF A DISCONTINUOUS GALERKIN METHOD FOR THE SIMULATION OF TURBULENT FLOW CONFIGURATIONS ON HYBRID MESHES

Marta de la Llave Plata[1], Florent Renac[2], Emeric Martin[3], Jean-Baptiste Chapelier[4] and Vincent Couaillier[5]

Onera, The French Aerospace Lab, F-92322 Châtillon, France, http://www.onera.fr/
[1]Marta.de_la_Llave_Plata@onera.fr, [2]Florent.Renac@onera.fr, [3]Emeric.Martin@onera.fr,
[4]Jean-Baptiste@Chapelier@onera.fr, [5]Vincent.Couaillier@onera.fr

Key Words: Discontinuous Galerkin method, hybrid mesh, turbulent flow, High-order, Discretization methods

Over the years, the development of new and increasingly powerful CFD simulation tools has helped manufacturers in the aerospace industry gain a greater understanding of the operating performance of their products. This has allowed them to progress through the design life cycle in a more timely and cost-effective manner by supplementing or replacing experimental testing with CFD computations. Today, the demand for very accurate CFD predictions at an ever-increasing level of detail is the driving force for the development of highly accurate simulation techniques able to predict not only overall flow features, but also local values of the quantities of interest.

Nevertheless, most of the industrial CFD codes used today are based on second-order methods able to predict only average quantities. Thus, Onera has started the development of a Discontinuous Galerkin (DG) solver called Aghora [1] [2] [3] with the aim of overcoming the limitations of second-order methods. The main goal is to develop a new demonstrator able to integrate efficient high-order schemes based on DG methods for the simulation of steady and unsteady turbulent flows on hybrid meshes (tetra, prisms, hex, …). Different levels of turbulent flow simulation, i.e. DNS, LES, hybrid RANS/LES and RANS are considered. Adaptive techniques based on local HPM methods (H for grid, P for accuracy of shape function, M for model) are also developed in order to represent accurately the physics of inhomogeneous flows, while keeping low the computational cost.

The solution in each element is expressed in terms of a polynomial expansion, the basis functions in two or three dimensions being built using tensor products of 1D polynomials. Jacobi polynomials are used for tetrahedral and parallelepiped meshes. A modified Gram-Schmidt orthonormalization procedure may also be used for general-shaped element grids. The Local Lax-Friedrichs flux or the Roe flux can be used to approximate the convective fluxes across the inter-element faces. The BR2 scheme [4] and the symmetric interior penalty methods are available to discretize the viscous terms. The semi-discrete equations can be advanced in time by means of explicit third- and fourth-order strong stability preserving Runge-Kutta methods or by Backward Euler implicit method.

In order to illustrate the benefits of using the high-order DG approach a number of applications have been performed with the Aghora solver in the framework of the European project IDIHOM. A number of several examples are presented hereafter.

Fig. 1 presents the evolution of the L^2-norm of the error vs. the number of degrees of freedom. The calculations are performed using the compressible Navier-Stokes equations for the simulation of the Poiseuille flow (M=0.1) for which an analytical solution is known (Method of Manufactured Solution). As expected, we can observe that the convergence rate is very close to the theoretical rate.

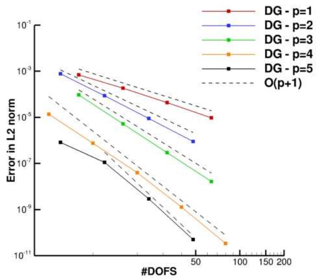

Figure 1: Convergence analysis in terms of DoFs – Manufactured solutions for Navier-Stokes equations Aghora (laminar flow) - L2-norm on the error between numerical and exact solutions

Fig. 2 presents two 3D Navier-Stokes computations, a jet exhausting in a fluid at rest on the left (JEAN nozzle configuration) and a transonic flow in a rotating compressor wheel on the right (NASA Rotor 37).

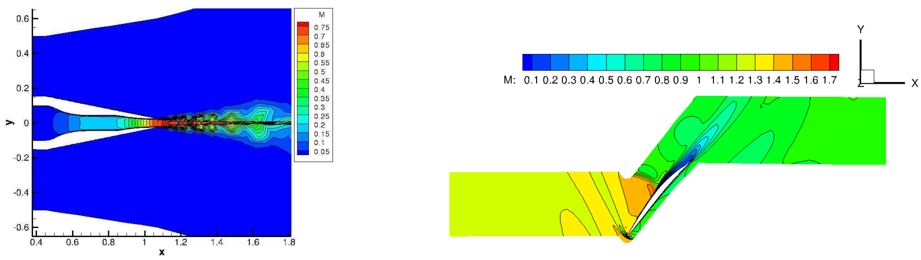

Fig. 2 : DG Navier-Stokes Computations using Aghora – Left : Jean Nozzle jet with P2 approximation (Mach number) – Right : Rotor 37 configuration with P1 approximation (Mach number)

References

[1] F. Renac, C. Marmignon and F. Coquel, Fast time implicit-explicit discontinuous Galerkin method for convection dominated flow problems, Math. Sci., 10 (2012), pp. 1161-1172

[2] J.-B. Chapelier, M. De La Llave Plata, and F. Renac, Inviscid and viscous simulations of the Taylor-Green vortex flow using a modal Discontinuous Galerkin approach, 42nd AIAA Fluid Dynamics Conference and Exhibit, June 2012, New Orleans, AIAA paper 2012-3073.

[3] F. Renac, S. Gerald, C. Marmignon and F. Coquel, Fast time implicit–explicit discontinuous Galerkin method for the compressible Navier–Stokes equations, J. Comp. Phy., Vol. 251 (2013), pp 272-291.

[4] F. Bassi and S. Rebay, Numerical evaluation of two discontinuous Galerkin methods for the compressible Navier-Stokes equations, Int. J. Num. Methods Fluids 40 (2002), pp. 197–202.