A stay at the Aerospace Computational Design Laboratory at the Massachusetts Institute of Technology (Cambridge-USA): developing discontinuous Galerkin methods for the resolution of the incompressible Navier-Stokes equations

Adeline de Villardi de Montlaur
Escola Politècnica Superior de Castelldefels
Universitat Politècnica de Catalunya
email: adeline.de.montlaur@upc.edu

I. Introduction

Aerodynamics is a branch of fluid dynamics concerned with the study of gas flows, first analyzed by George Cayley in the 1800s. The solution of an aerodynamic problem normally involves calculating for various properties of the flow, such as velocity, pressure, density, and temperature, as a function of space and time. Understanding the flow pattern makes it possible to calculate or approximate the forces and moments acting on bodies in the flow. If one of the major goals of aerodynamics is to predict the aerodynamic forces on an aircraft it is also a significant factor in any type of vehicle design, including automobiles, as well as in the prediction of forces and moments in sailing. Civil engineers also use aerodynamics, and particularly aeroelasticity, to calculate wind loads in the design of large buildings and bridges.

There are two main ways of studying the aerodynamic of a vehicle: an experimental one, using for example wind tunnel and the numerical one, using the so called Computational Fluid Dynamics (CFD). It is the science of predicting fluid flow, heat and mass transfer, chemical reactions, and related phenomena by solving numerically the set of governing mathematical equations. A discretization of the spatial domain into small cells to form a volume mesh is needed, and then a suitable algorithm is applied to solve the equations of motion. The results of CFD analysis are relevant in conceptual studies of new designs and detailed product development as well as troubleshooting and redesign. It complements testing and experimentation of a new design, reducing the total effort required in the experiment design and data acquisition.

Aerodynamic problems can be classified in a number of ways. The flow environment defines the first classification criterion. External aerodynamics is the study of flow around solid objects of various shapes, for example the study of a wing profile, whereas internal aerodynamics is the study of flow through passages in solid objects, for example the study of the internal flow of a wind tunnel, especially of the boundary layers near its walls. The ratio of the problem's characteristic flow speed to the speed of sound comprises a second classification of aerodynamic problems. For example a problem is called subsonic if all the speeds in the problem are less than the speed of sound. The influence of viscosity in the flow dictates a third classification. Some problems involve only negligible viscous effects on the solution, in which case viscosity can be considered to be nonexistent, whereas flows for which viscosity cannot be neglected are called viscous flows.

The mathematical equations describing the fluid dynamic depend on the nature of the problem. The Navier-Stokes equations, named after Claude-Louis Navier and George Gabriel Stokes, are a set of equations that establish that changes in momentum of the particles of a fluid are simply the product of changes in pressure and dissipative viscous forces acting inside the fluid. These viscous forces originate in molecular interactions and dictate how viscous a fluid is. Thus, the Navier-Stokes equations are a dynamical statement of the balance of forces acting at any given region of the fluid. They are one of the most useful sets of equations because they describe the physics of a large number of phenomena like ocean currents, water flow in a pipe, flow around an airfoil, etc.

The context of my PhD thesis is to study the flow of an incompressible viscous fluid, by solving the incompressible Navier-Stokes equations in an efficient way. At the beginning of my stay at the Aerospace Computational Design Laboratory at the Massachusetts Institute of Technology (Cambridge-USA) I had already developed a method to solve the Stokes equations. The following report shows how I applied this method to the incompressible Navier-Stokes equations, how I applied an algorithm used at MIT to these equations, and how I used different types of elements to solve these equations.

II. The Navier-Stokes equations: interior penalty method and compact discontinuous Galerkin method

In the proposed method, the incompressible Navier-Stokes equations are solved using an exactly divergence-free velocity approximation. Let Ω be an open, bounded domain with piecewise linear boundary $\partial\Omega$. Suppose that Ω is partitioned in n disjoint subdomains K, which for example correspond to different materials, with also piecewise linear boundaries ∂K that define an internal interphase Γ. The strong form for the homogeneous steady Navier-Stokes problem can be written as:

$$\begin{aligned} -\nu\boldsymbol{\nabla}^2\boldsymbol{u} + \nabla p + \boldsymbol{\nabla}\cdot(\boldsymbol{u}\otimes\boldsymbol{u}) &= \boldsymbol{f} && \text{in } \Omega, \\ \boldsymbol{\nabla}\cdot\boldsymbol{u} &= 0 && \text{in } \Omega, \\ \boldsymbol{u} &= \boldsymbol{u}_D && \text{on } \Gamma_D, \\ \boldsymbol{n}\cdot\boldsymbol{\sigma} &= \boldsymbol{t} && \text{on } \Gamma_N, \end{aligned}$$

where $\boldsymbol{f}$ is a source term, $\boldsymbol{u}$ the flux velocity, p its pressure, and ν the kinematic viscosity.

Here a discontinuous Galerkin method for incompressible flows is used [Reed]. Divergence-free piecewise polynomial velocity spaces are used, yielding an exactly divergence-free solution. Considering the Stokes and Navier-Stokes equations, a hybrid method is used where the original problem can be split into two uncoupled problems. The first one solves for the velocity and the hybrid pressures (pressures along the mesh edges) [Carrero]. This discrete system has fewer degrees of freedom than the original mixed method since the pressures are defined just on the mesh edges. It can be further reduced using a non-consistent penalty method that uncouples the calculation of the velocity and the hybrid pressures. The second part of the problem, which requires the solution of the previous one, evaluates the interior pressures.

Two methods have been studied here. In one case, in order to enforce continuity between elements when using discontinuous piecewise polynomials an interior penalty method is used. Penalty terms are used on each edge, penalizing the jump of the function across the edge, and providing stability [Arnold], [Babuska].

In the other case and in order to avoid the choice of an arbitrary penalty term, the so-called compact discontinuous Galerkin method is used [Peraire], [Cockburn].

III. Raviart and Thomas elements

Toward the end of the stay at the Aerospace Computational Design Laboratory at the Massachusetts Institute of Technology, I have started to work on another type of elements: the Raviart and Thomas elements. Combining the use of triangle and rectangle elements should allow to use a mixed formulation in order to use rectangle when the geometry is regular and triangles where the geometry is more complex or for the boundaries of the object of study.

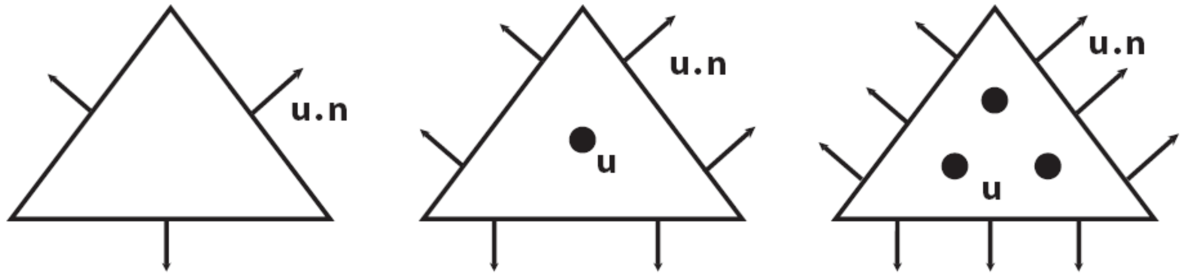

Figure 1: Raviart and Thomas triangle elements: from left to right: RT_0, RT_1, RT_2.

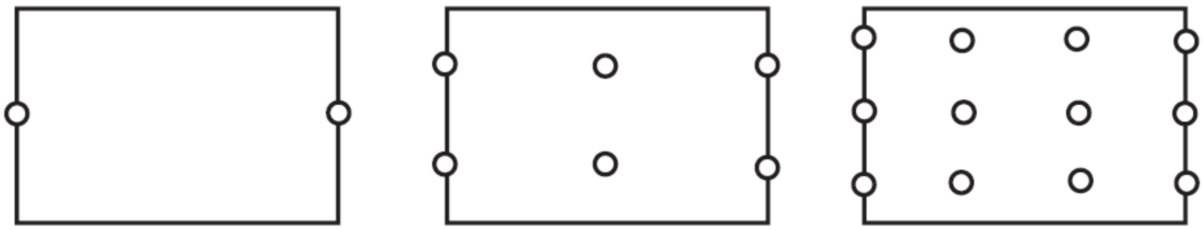

Figure 2: Raviart and Thomas rectangle elements: from left to right: RT_0, RT_1, RT_2. Only the nodes for the x-component of the velocity are represented, for the y-component of the velocity the nodes are placed in a similar way (switching the x-axis for the y-axis).

IV. Numerical examples

In order to validate the numerical methods developed, it is necessary to apply them to some basic numerical examples before using them on a more complex, and realistic, example. A standard benchmark test for incompressible flows is to model a plane flow of an isothermal fluid in a square lid-driven cavity; it has a zero body force and one moving wall, here the upper wall. Figure 3 shows the results of the velocity streamlines and the field pressure for two different Reynolds number. The effect of the increase in the Reynolds number is seen as the center of the velocity vortex is displaced. The pressure shows infinite values at the two upper corners due to the discontinuity of the velocity at these particular points.

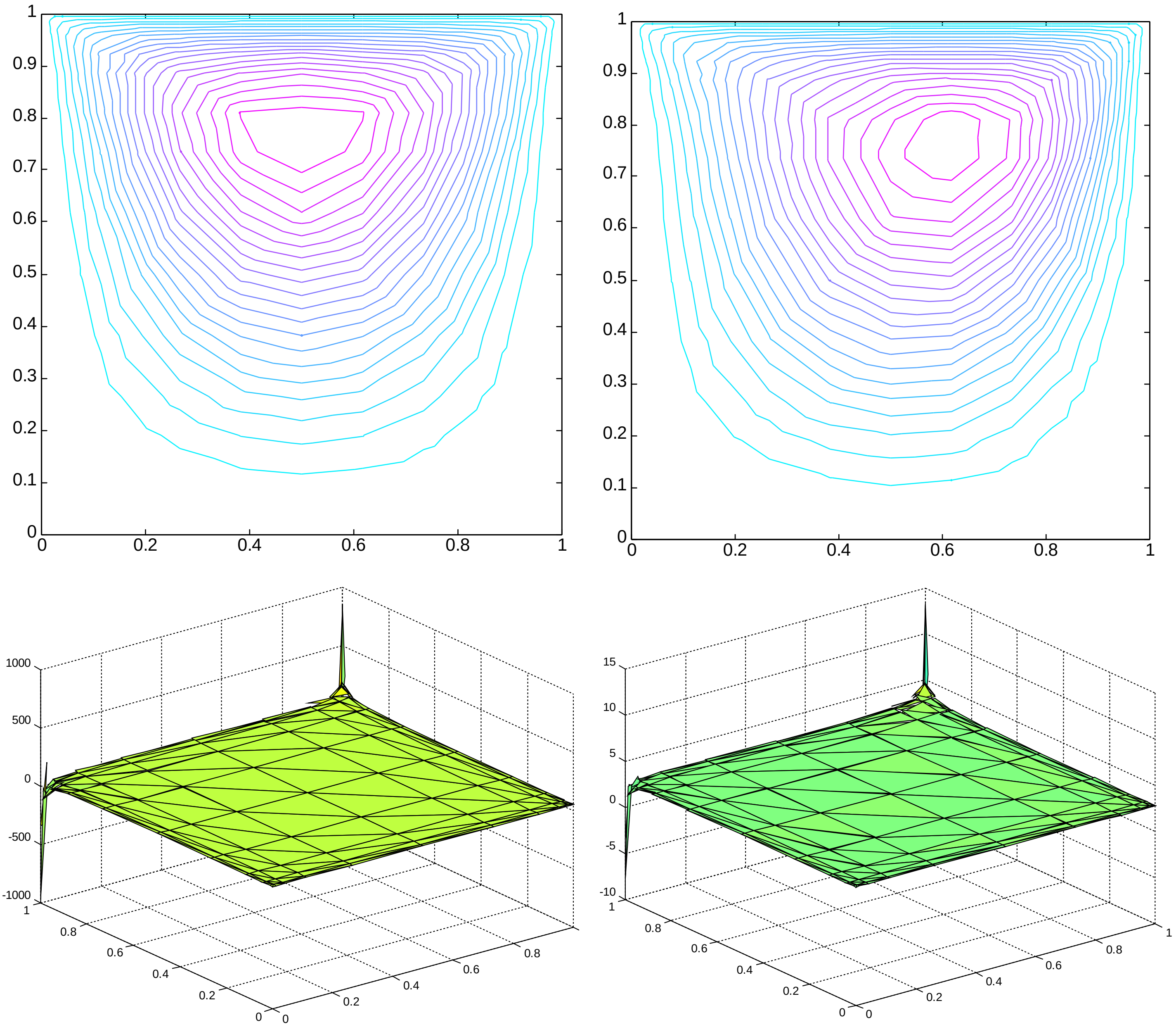

Figure 3: Driven cavity example: velocity streamlines (top) and pressure field (bottom) for a Reynolds number = 1 (left) and 100 (right)

Another type of application is a flow in an idealized porous medium [Okkels]. A fluid flowing in an idealized porous medium is subject to a friction force proportional to the fluid velocity $\boldsymbol{u}$. This kind of problem is derived from the Stokes equations and is called the Darcy's law. It is valid for slow, viscous flow, like for example groundwater flows.

Figure 4 represents the velocity vectors within a porous domain: the grey part represents a porous material, the white ones an empty domain.

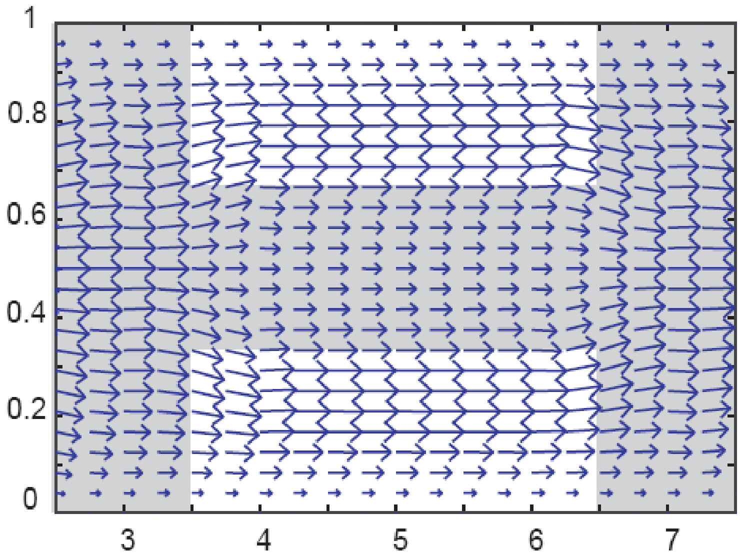

Figure 4: Velocity vectors within a porous domain

It can be seen that the empty regions divert the flow away from the center of the channel; the flow tends to go into the empty domains where it is more accelerated than in the porous one.

V. Conclusion

My stay at the Aerospace Computational Design Laboratory at the Massachusetts Institute of Technology (Cambridge-USA) has been very proficient. It has indeed allowed me to broaden my knowledge in numerical methods, allowing me to work with researchers specialized in this field and its application to aerospace engineering. At MIT I also had the opportunity to attend seminars and workshops in the aerospace field, and more particularly in computational design.

References

[Peraire] J. Peraire and P.O. Persson, A compact discontinuous Galerkin (CDG) method for elliptic problems, in review. 2007

[Arnold] D. N. Arnold. An interior penalty finite element method with discontinuous elements. *SIAM Journal on Numerical Analysis*, 19, p. 742-760.1982.

[Okkels] Okkels F, Olesen LH, Bruus H. Applications of Topology Optimization in the Design of Micro- and Nanofluidic Systems. In *Technical Proceedings of the 2005 NSTI Nanotechnology Conference and Trade Show*, p. 575-578, 2005.

[Reed] Reed WH, Hill TR. Triangular mesh methods for the neutron transport equation. In *Technical Report* 1973, LA-UR-73-479, Los Alamos Scientific Laboratory.

[Carrero] Carrero J, Cockburn B, Schotzau D. Hybridized Globally Divergence-free LDG methods. *Mathematics of Computation*, 75, p. 533-563, 2005.

[Cockburn] Cockburn B, Li F, Shu C. Locally divergence-free discontinuous Galerkin methods for the Maxwell equations. *Journal of Computational Physics*, 194, p. 588-610, 2004.

[Babuska] Babuska I. The finite element method with penalty. *Mathematics of Computation*, 27, p.221-228, 1973.