A HIGH-ORDER DISCONTINUOUS GALERKIN METHOD FOR NATURAL CONVECTION PROBLEMS

F. Bassi, A. Crivellini

Dipartimento di Ingegneria Industriale
Università di Bergamo
viale Marconi 5, 24044 Dalmine (BG), Italy
e-mail: francesco.bassi@unibg.it, andrea.crivellini@unibg.it

Key words: Incompressible flow, Navier-Stokes equations, Discontinuous Galerkin method, Artificial compressibility, Riemann solver, Boussinesq approximation

Abstract. Discontinuous Galerkin (DG) methods have proved to be very well suited for the construction of robust high-order numerical schemes on unstructured and possibly non conforming grids for a wide variety of problems. In this paper we consider natural convection flow problems and present a high-order DG method for their numerical solution. The governing equations are the incompressible Navier-Stokes (INS) with the Boussinesq approximation to represent buoyancy effects and the energy equation to describe the temperature field.

The method here presented is an extension to natural convection flows of a novel high-order DG method for the numerical solution of the INS equations, recently proposed in Bassi et al.\textsuperscript{1}. The distinguishing feature of this method is the formulation of the inviscid interface flux which is based on the solution of local Riemann problems associated with the artificial compressibility perturbation of the incompressible Euler equations. The discretization of the viscous term follows the well established DG scheme named BR\textsuperscript{2,3}. The method is fully implicit and the solution is advanced in time using either a first order backward Euler or a second order Runge-Kutta scheme.

To assess the capabilities of the DG method presented in this paper we computed second-, third- and fourth-order space-accurate solutions of several benchmark problems on natural convection in two-dimensional cavities.

1 INTRODUCTION

Discontinuous Galerkin (DG) methods have proved to be suited for the construction of robust high-order numerical schemes on arbitrary unstructured and possibly non conforming grids for a wide variety of problems. The application of the DG space discretization to incompressible flows has been considered in a few recent works. Liu and Shu\textsuperscript{4} introduced a DG method for 2D incompressible flows in stream function formulation and
in a series of papers\cite{Cockburn2000, Cockburn2002, Cockburn2003} Cockburn and coworkers proposed and thoroughly analyzed the Local Discontinuous Galerkin (LDG) method for the Stokes, Oseen and Navier-Stokes equations.

One of the key ingredients of DG methods is the formulation of interface (numerical) fluxes, which provide weak coupling of discontinuous solutions in neighboring elements. In the inviscid compressible case the numerical flux is often computed by exploiting the hyperbolic nature of the equations as the (approximate or exact) solution of a Riemann problem. In the incompressible case, however, the equations are no longer hyperbolic due to the lack of the unsteady term in the continuity equation and it is therefore not possible to compute the interface flux following the same approach.

In this paper we continue the development of the DG method for the INS equations introduced in Bassi et al.\cite{Bassi2000}. The novelty of this method is the formulation of the inviscid numerical flux which is based on the solution of the Riemann problem for the incompressible Euler equations with a relaxed incompressibility constraint. It is important to remark that the idea of relaxing the incompressibility constraint by an artificial compressibility term is employed only for the construction of the interface fluxes.

The numerical experiments reported in Bassi et al.\cite{Bassi2000} show that the convergence rate, using polynomials of degree $k$ for all the variables, is $k + 1$ for the velocity components and at least $k$ for the pressure. In that paper it is also shown that the method is well suited for INS computations at high Reynolds number.

Here the method is extended to solve the INS equations coupled with the energy conservation equation using the Boussinesq approximation in the body force term of the momentum equation. This set of equations is a suitable physical model to describe many natural convection problems. In the following sections we will show that the treatment of the additional energy equation in the Riemann problem is straightforward.

As regards the DG space discretization of the diffusive terms we have used the well established DG scheme introduced in Bassi et al.\cite{Bassi2001}. Finally, the space discretized equations are integrated in time by means of fully implicit first and second order accurate methods and the resulting linear systems are solved using LU direct or GMRES iterative solvers.

The DG method here developed has been validated by computing two steady\cite{Bassi2002, Bassi2003, Bassi2004} (with linear and non linear buoyancy effects) and one unsteady\cite{Bassi2002, Bassi2003, Bassi2004} “thermally driven cavity flow” problems.

\section{GOVERNING EQUATIONS}

We consider the conservation equations of mass, momentum and energy for an incompressible fluid. Following the Boussinesq approximation, buoyancy effects are accounted for by introducing a variable density body force term in the momentum equation. The density in this term depends on the temperature according to the equation

$$
\rho(T) = \rho_r(1 - \beta \Delta T),
$$

\section{NUMERICAL RESULTS}

The results presented in this section are obtained by solving the governing equations with the DG method for the INS equations...
where $\rho$ is the density, $\rho_r$ is the reference density, $\Delta T$ is the difference between the temperature and its reference value and $\beta$ is the thermal expansion coefficient.

The non dimensional form of the governing equations is obtained assuming a length scale $L_r$, a temperature difference scale $\Delta T_r$ and the buoyancy velocity scale $U_r = \sqrt{gL_r \beta \Delta T_r}$, where $g$ is the acceleration of gravity.

With the above choices the governing equations read:

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

$$\frac{\partial \mathbf{u}}{\partial t} + \nabla \cdot \mathbf{F}(\mathbf{u}, p) - \nabla \cdot \left( \frac{\text{Pr}}{\sqrt{\text{Ra} \cdot \text{Pr}}} \nabla \mathbf{u} \right) = \theta \mathbf{j},$$

$$\frac{\partial \theta}{\partial t} + \nabla \cdot \mathbf{g}(\mathbf{u}, \theta) - \nabla \cdot \left( \frac{1}{\sqrt{\text{Ra} \cdot \text{Pr}}} \nabla \theta \right) = 0,$$

in $[0, t] \times \Omega$, $\Omega \subset \mathbb{R}^N$, where $N \in \{2, 3\}$ is the number of space dimensions, $\mathbf{j}$ is the unit vector in the $y$ direction, $\mathbf{u}$ and $\theta$ denote the dimensionless velocity vector and temperature difference, $\text{Pr}$ is the Prandtl number and

$$\text{Ra} = \text{Pr} \frac{gL_r^3 \beta \Delta T_r}{\nu^2},$$

the Rayleigh number ($\nu$ is the kinematic viscosity), is the main parameter characterizing the flow field.

The convective terms of the governing equations are written in conservation form because this is the natural starting point for the inviscid flux discretization described in the following section. The fluxes $\mathbf{F}(\mathbf{u}, p)$ and $\mathbf{g}(\mathbf{u}, \theta)$ are given by

$$\mathbf{F}(\mathbf{u}, p) \overset{\text{def}}{=} \mathbf{u} \otimes \mathbf{u} + p \mathbf{I} = u_i u_j + p \delta_{ij},$$

$$\mathbf{g}(\mathbf{u}, \theta) \overset{\text{def}}{=} \mathbf{u} \theta = u_i \theta,$$

where $p$ is the dimensionless deviation from hydrostatic pressure and $\mathbf{I}$ is the identity matrix.

3 DISCONTINUOUS GALERKIN DISCRETIZATION

In this section the DG discretization of the governing equations is first introduced for the inviscid part of Eq. (2), and then completed with the discretization of the viscous terms. The inviscid and viscous forms of Eq. (2) are loosely denoted as Euler and Navier-Stokes equations.
3.1 Euler equations

Neglecting the diffusive terms, the weak form of Eq. (2) reads:

\[- \int_\Omega \nabla q \cdot u \, dx + \int_{\partial \Omega} q u \cdot n \, d\sigma = 0,\]

\[\int_\Omega v \cdot \frac{\partial u}{\partial t} \, dx - \int_\Omega \nabla v : F(u, p) \, dx + \int_{\partial \Omega} v \otimes n : F(u, p) \, d\sigma = \int_\Omega v \cdot j \theta \, dx,
\]

\[\int_\Omega q \cdot g(u, \theta) \, dx + \int_{\partial \Omega} q g(u, \theta) \cdot n \, d\sigma = 0,\]

for arbitrary test functions \(v\) and \(q\).

In order to construct a DG discretization of Eq. (3), we consider a triangulation \(T_h = \{K\}\) of an approximation \(\Omega_h\) of \(\Omega\), that is we partition \(\Omega_h\) into a set of non-overlapping elements \(K\) (not necessarily simplices). We denote with \(\mathcal{E}^0_h\) the set of internal element faces, with \(\mathcal{E}^\partial_h\) the set of boundary element faces and with \(\mathcal{E}_h = \mathcal{E}^0_h \cup \mathcal{E}^\partial_h\) their union. We moreover set

\[\Gamma^0_h = \bigcup_{e \in \mathcal{E}^0_h} e, \quad \Gamma^\partial_h = \bigcup_{e \in \mathcal{E}^\partial_h} e, \quad \Gamma_h = \Gamma^0_h \cup \Gamma^\partial_h.\]

The solution is approximated on \(T_h\) as a piecewise polynomial function possibly discontinuous on element interfaces, i.e. we assume the following space settings for the approximate solution \(u_h, p_h\) and \(\theta_h\):

\[u_h \in V_h \overset{\text{def}}{=} [V_h]^N, \quad p_h, \theta_h \in Q_h \overset{\text{def}}{=} V_h,\]

with

\[V_h \overset{\text{def}}{=} \left\{ v_h \in L^2(\Omega) : v_h|_K \in P_k(K) \, \forall K \in T_h \right\}\]

for some polynomial order \(k \geq 1\), being \(P_k(K)\) the space of polynomials of global degree at most \(k\) on the element \(K\).

In order to simplify the presentation, it is convenient to introduce some trace operators which generalize those defined in Arnold et al.\(^3\). On a generic internal face \(e \in \mathcal{E}^0_h\), see
Figure 1, we define
\[ [\mathbf{v}] \overset{\text{def}}{=} \mathbf{v}^+ \otimes \mathbf{n}^+ + \mathbf{v}^- \otimes \mathbf{n}^-, \quad [q] \overset{\text{def}}{=} q^+ \mathbf{n}^+ + q^- \mathbf{n}^- , \] (6)
where \( \mathbf{v} \) denotes a generic vector quantity and \( q \) a generic scalar quantity. Notice that
\[ [\mathbf{v}] \] is a tensor quantity, and \([q] \) is a vector quantity, \( \text{i.e.} \) this operator always increases the
tensor rank by one.

Moreover, we introduce the average operator
\[ \{ \cdot \} \overset{\text{def}}{=} \frac{(\cdot)^+ + (\cdot)^-}{2} , \] (7)
which applies to scalars, vectors or tensors. These definitions can be suitably extended to
facing intersecting \( \partial \Omega \) accounting for the weak imposition boundary conditions as shown
in \( \S \ 3.3 \).

The discrete counterpart of Eq. (3) for a generic element \( K \in T_h \) then reads:
\[
- \int_K \nabla_h q_h \cdot \mathbf{u}_h \, dx + \int_{\partial K} q_h \mathbf{u}_h|_K \cdot \mathbf{n} \, d\sigma = 0 , \\
\int_K \mathbf{v}_h \cdot \frac{\partial \mathbf{u}_h}{\partial t} \, dx - \int_K \nabla_h \mathbf{v}_h : \mathbf{F}(\mathbf{u}_h, p_h) \, dx \\
+ \int_{\partial K} \mathbf{v}_h \otimes \mathbf{n} : \mathbf{F}(\mathbf{u}_h|_K, p_h|_K) \, d\sigma = \int_K \mathbf{v}_h \cdot \mathbf{j}_h \, dx , \\
\int_K q_h \frac{\partial \theta_h}{\partial t} \, dx - \int_K \nabla_h q_h \cdot \mathbf{g}(\mathbf{u}_h, \theta_h) \, dx + \int_{\partial K} q_h \mathbf{g}(\mathbf{u}_h|_K, \theta_h|_K) \cdot \mathbf{n} \, d\sigma = 0 .
\] (8)

To introduce a coupling between the degrees of freedom belonging to adjacent elements
and to ensure conservation, we substitute the fluxes \( \mathbf{u}_h|_K, \mathbf{F}(\mathbf{u}_h|_K, p_h|_K) \) and \( \mathbf{g}(\mathbf{u}_h|_K, \theta_h|_K) \) with suitably defined numerical fluxes \( \hat{\mathbf{u}}(\mathbf{u}_h^+, p_h^+), \hat{\mathbf{F}}(\mathbf{u}_h^+, p_h^+) \) and \( \hat{\mathbf{g}}(\mathbf{u}_h^+, p_h^+, \theta_h^+) \). We remark
that the stability and accuracy properties of the method strongly depend on the choice of
such numerical fluxes.

Summing Eq. (8) over the elements we obtain the DG formulation of problem (3) which
then requires to find \( \mathbf{u}_h \in V_h \) and \( p_h, \theta_h \in Q_h \) such that
\[
- \int_{\Omega_h} \nabla_h q_h \cdot \mathbf{u}_h \, dx + \int_{\Gamma_h} \{q_h\} : \hat{\mathbf{u}}(\mathbf{u}_h^+, p_h^+) \, d\sigma = 0 , \\
\int_{\Omega_h} \mathbf{v}_h \cdot \frac{\partial \mathbf{u}_h}{\partial t} \, dx - \int_{\Omega_h} \nabla_h \mathbf{v}_h : \mathbf{F}(\mathbf{u}_h, p_h) \, dx \\
+ \int_{\Gamma_h} \{\mathbf{v}_h\} : \hat{\mathbf{F}}(\mathbf{u}_h^+, p_h^+) \, d\sigma = \int_{\Omega_h} \mathbf{v}_h \cdot \mathbf{j}_h \, dx , \\
\int_{\Omega_h} q_h \frac{\partial \theta_h}{\partial t} \, dx - \int_{\Omega_h} \nabla_h q_h \cdot \mathbf{g}(\mathbf{u}_h, \theta_h) \, dx + \int_{\Gamma_h} \{q_h\} : \hat{\mathbf{g}}(\mathbf{u}_h^+, p_h^+, \theta_h^+) \, d\sigma = 0 ,
\] (9)
for all $v_h \in V_h$ and $q_h \in Q_h$.

The key idea adopted to compute $\mathbf{\hat{u}}$, $\mathbf{\hat{F}}$ and $\mathbf{\hat{g}}$ is to reduce the problem of flux computation to the solution of a planar Riemann problem as in the compressible case. In order to recover the hyperbolic character of the equations, the incompressibility constraint is relaxed by adding an artificial compressibility term to the continuity equation. At each quadrature point $P$ on $\Gamma_h$ we therefore solve the Riemann problem for the equations

\begin{alignat}{2}
\frac{1}{c^2} \frac{\partial p}{\partial t} + \frac{\partial u}{\partial x} &= 0, \\
\frac{\partial u}{\partial t} + \frac{1}{x} \frac{\partial (u^2 + p)}{\partial x} &= 0,
\end{alignat}

(10)

with initial datum

$$(u, p, \theta) = \begin{cases}
(u_h^-, p_h^-, \theta_h^-) & \text{if } x < 0, \\
(u_h^+, p_h^+, \theta_h^+) & \text{if } x > 0,
\end{cases}$$

where $x$ denotes a locally defined axis oriented as the normal vector $\mathbf{n}^+$ pointing out of $K^+$ and located in such a way that $x = 0$ at $P$, see Figure 1.

Denoting with $(u_s, p_s, \theta_s)$ the solution of the Riemann problem on the space-time line $x/t = 0$, we finally set

$$\mathbf{\hat{u}}(u_h^\pm, p_h^\pm) = u_s, \quad \mathbf{\hat{F}}(u_h^\pm, p_h^\pm) = F(u_s, p_s), \quad \mathbf{\hat{g}}(u_h^\pm, p_h^\pm, \theta_h^\pm) = g(u_s, \theta_s).$$

It is worth noticing that, unlike in Cockburn et al.\textsuperscript{5-7}, we do not split the numerical flux related to the advection term from that related to the pressure term, thus establishing a stronger link between pressure and velocity which seems to enhance the properties of the method. Due to space limitations, the procedure for the determination of the state $(u_s, p_s, \theta_s)$ will not be reported here. We refer to Bassi et al.\textsuperscript{1} for a thorough description of the solution of the Riemann problem for $(u_s, p_s)$. As regards the solution of the additional energy equation in system (10), we simply observe that the equation for $\theta$ has exactly the same form as that for the tangential velocity component $v$. Hence, the form of the solution for $v_s$ reported in Bassi et al.\textsuperscript{1} holds also for $\theta_s$.

### 3.2 Navier-Stokes equations

Many techniques are available for the DG space discretization of the diffusive terms: a complete survey can be found in Arnold et al.\textsuperscript{3} In this paper we choose the form, first proposed in Bassi et al.\textsuperscript{2}, which gives the following space discretization of the complete
governing equations: find \( u_h \in V_h \) and \( p_h, \theta_h \in Q_h \) such that

\[
- \int_{\Omega_h} \nabla_h q_h \cdot u_h \, dx + \int_{\Gamma_h} \left[ q_h \right] \cdot \tilde{u}(u_h^+, p_h^+) \, d\sigma = 0,
\]

\[
\int_{\Omega_h} v_h \cdot \frac{\partial u_h}{\partial t} \, dx - \int_{\Omega_h} \nabla_h v_h : (F_v(\nabla_h u_h, r) + F(u_h, p_h)) \, dx
+ \int_{\Gamma_h} \left[ v_h \right] : \left( \tilde{F}_v(\nabla_h u_h^+, r_e^+) + \tilde{F}(u_h^+, p_h^+) \right) \, d\sigma = \int_{\Omega_h} v_h \cdot j_h \, dx,
\] (11)

\[
\int_{\Omega_h} q_h \frac{\partial \theta_h}{\partial t} \, dx - \int_{\Omega_h} \nabla_h q_h \cdot (g_v(\nabla_h \theta_h, r) + g(u_h, \theta_h)) \, dx
+ \int_{\Gamma_h} \left[ q_h \right] \cdot \left( \tilde{g}_v(\nabla_h \theta_h^+, r_e^+) + \tilde{g}(u_h^+, p_h^+, \theta_h^+) \right) \, d\sigma = 0,
\]

for all \( v_h \in V_h \) and \( q_h \in Q_h \). The diffusive fluxes in the above equations are defined as:

\[
F_v(\nabla_h u_h, r) \overset{\text{def}}{=} -\frac{Pr}{\sqrt{Ra} \cdot Pr} (\nabla_h u_h + r(\left[ u_h \right])),
\]

\[
\tilde{F}_v(\nabla_h u_h^+, r_e^+) \overset{\text{def}}{=} -\frac{Pr}{\sqrt{Ra} \cdot Pr} \left( \left\{ \nabla_h u_h \right\} + \eta_e \left\{ r_e(\left[ u_h \right]) \right\} \right),
\]

\[
ge_v(\nabla_h \theta_h, r) \overset{\text{def}}{=} -\frac{1}{\sqrt{Ra} \cdot Pr} (\nabla_h \theta_h + r(\left[ \theta_h \right])),
\]

\[
\tilde{g}_v(\nabla_h \theta_h^+, r_e^+) \overset{\text{def}}{=} -\frac{1}{\sqrt{Ra} \cdot Pr} \left( \left\{ \nabla_h \theta_h \right\} + \eta_e \left\{ r_e(\left[ \theta_h \right]) \right\} \right),
\] (12)

where \( r_e \) is the lifting operator, that can be defined for the jumps of both vector and scalar quantities, and \( r \) is given by

\[
r(\cdot) \overset{\text{def}}{=} \sum_{e \in \mathcal{E}_h} r_e(\cdot),
\] (13)

being \( e \) a generic face. When applied to the jump of the velocity vector \( u, r_e \) is defined as the solution of the following problem:

\[
\int_{\Omega_h} r_e(\psi) : \tau_h \, dx = -\int_{\Omega_h} \psi : \left\{ \tau_h \right\} \, d\sigma, \quad \forall \tau_h \in [V_h]^N, \quad \psi \in \left[ L^1(e) \right]^N^2,
\] (14)

whereas applied to the jump of the scalar quantity \( \theta \) it is defined as:

\[
\int_{\Omega_h} r_e(w) \cdot v_h \, dx = -\int_{\Omega_h} w \cdot \left\{ v_h \right\} \, d\sigma, \quad \forall v_h \in [V_h]^N, \quad w \in \left[ L^1(e) \right]^N.
\] (15)

It is possible to find lower bounds for the parameter \( \eta_e \in \mathbb{R}^+ \) ensuring stability of the method. We refer the interested reader to Arnold et al.\textsuperscript{3} for the details.
3.3 Boundary conditions

The DG discretization is best suited for a weak enforcement of boundary conditions. This can easily be achieved by properly defining a boundary state which, together with the internal state, allows to compute the numerical fluxes and the lifting operator on the portion $\Gamma^B_h$ of the boundary $\Gamma_h$.

The wall-type boundary conditions have been implemented by defining the boundary state on the exterior of boundary faces as $u^b_h = -u^+_h$, $\nabla u^b_h = \nabla u^+_h$ and $p^b_h = p^+_h$. Dirichlet boundary conditions for $\theta$ prescribe $\theta^b_h$ and set $\nabla \theta^b_h = \nabla \theta^+_h$, while for adiabatic walls we set $\theta^b_h = \theta^+_h$ and $\nabla \theta^b_h = -\nabla \theta^+_h$. For all these cases the external boundary state exactly replaces $u^b_h$, $\theta^b_h$ and $p^b_h$ in the jump operators, in the numerical fluxes and in the lifting operator.

3.4 Time discretization and linear system solution

All integrals appearing in Eq. (11) are computed by means of Gauss integration rules with a number of integration points suited for the required accuracy. Cheaper non-product formulae are preferred to tensor product ones when available. The quadrature formulae are taken from the encyclopaedia of cubature formulae developed and maintained by Cools.\textsuperscript{12}

The discrete problem corresponding to Eq. (11) can be written as:

\[
M \frac{dW}{dt} + R(W) = 0, \tag{16}
\]

where $W$ is the global vector of unknown degrees of freedom and $M$ is the global block diagonal mass matrix. Lacking the time derivative of pressure in the governing equations, the blocks of $M$ corresponding to the pressure degrees of freedom are identically zero. Eq. (16) defines a system of (nonlinear) ODEs which has been discretized by means of the second-order implicit Runge-Kutta scheme proposed by Iannelli and Baker.\textsuperscript{13}

\[
W^{n+1} - W^n = Y_1 K_1 + Y_2 K_2,
\]

\[
\begin{bmatrix}
\frac{M}{\Delta t} + \alpha \frac{\partial R(W^n)}{\partial W}
\end{bmatrix} K_1 = -R(W^n),
\]

\[
\begin{bmatrix}
\frac{M}{\Delta t} + \alpha \frac{\partial R(W^n)}{\partial W}
\end{bmatrix} K_2 = -R(W^n + \beta K_1),
\tag{17}
\]

where $\partial R(W^n)/\partial W$ is the Jacobian matrix of the DG space discretization and the constants $\alpha$, $\beta$, $Y_1$, $Y_2$ are given by

\[
\alpha = \frac{2 - \sqrt{2}}{2}, \quad \beta = 8 \alpha \left( \frac{1}{2} - \alpha \right), \quad Y_1 = 1 - \frac{1}{8\alpha}, \quad Y_2 = 1 - Y_1.
\]

Notice that the one-step backward Euler and Crank-Nicolson schemes can be obtained by setting $Y_1 = 1$, $Y_2 = 0$, $\alpha = 1$ or $Y_1 = 1$, $Y_2 = 0$, $\alpha = 1/2$, respectively. Each step in
Eq. (17) requires to solve a linear system of the form $A x + b = 0$. However, since the matrix $A$ is the same for the two steps, the Jacobian matrix needs to be evaluated only once.

The matrix $A$ can be regarded as an $N_K \times N_K$ block sparse matrix where $N_K$ is the number of elements in $T_h$ and the rank of each block is $N_{DOF}^K \times (N + 2)$, being $N_{DOF}^K$ the number of degrees of freedom for each variable in the generic element $K$. Thanks to the DG discretization here adopted the degrees of freedom of a generic element $K$ are only coupled with those of the neighbouring elements and the number of nonzero blocks for each (block) row $K$ of the matrix $A$ is therefore equal to the number of elements surrounding the element $K$ plus one.

The Jacobian matrix of the DG discretization has been computed analytically without any approximation and, using very large time steps, the method can therefore achieve quadratic convergence in the computation of steady state solutions. For the backward Euler scheme and in the limit $\Delta t \rightarrow \infty$ Eq. (17) is in fact identical to one iteration of the Newton method applied to the steady discrete problem.

To solve Eq. (17) our code can use one of the numerous approaches (direct or iterative, sequential or parallel) available in the PETSc library (Portable Extensible Toolkit for Scientific Computations), the software upon which our DG code relies for the purpose of parallelization. The parallelization is based on grid partitioning accomplished by means of the METIS package. In this context each processor owns the data related to its local portion of the grid and the data on remote processors are accessed through MPI, the standard for message-passing communication. Thanks to the compactness of our DG method only the data owned by the near neighbor elements at partition boundaries need to be shared among different processes.

4 NUMERICAL RESULTS

In this section we consider three benchmark problems of natural convection flow within enclosed rectangular cavities. The boundary conditions prescribe the temperature on the vertical walls and no heat flux on top and bottom walls. In all cases the gravity force is directed downwards.

4.1 First benchmark problem

The first benchmark problem is the well known steady “thermally driven cavity flow”. The flow domain is a square $(0,1)^2$ filled with a Boussinesq fluid (air with $Pr = 0.71$).

The problem, with Rayleigh numbers up to $10^6$, was selected as the test case of a numerical benchmark in 1983. More recently several researchers extended the benchmark including the $Ra = 10^7$ and $Ra = 10^8$ cases. Being very close to the critical Rayleigh number beyond which the flow becomes unsteady ($(1.82 \pm 0.01) \cdot 10^8$, according to Le Quéré et al.), the $Ra = 10^8$ case represents a severe test for any numerical method.

Here we present the numerical results for the $Ra = 10^7$ and $Ra = 10^8$ cases. For
each Rayleigh number we have computed $P_1$, $P_2$ and $P_3$ solutions on two structured grids with $40 \times 40$ and $80 \times 80$ quad elements, clustered near the walls. The maximum grid spacing, $h_{\text{max}} = 0.045$, is the same for the two grids, whilst the minimum grid spacing is $h_{\text{min}} = 0.012$ for the coarse grid and $h_{\text{min}} = 0.0012$ for the fine grid. The results here reported have been computed with the artificial compressibility parameter $c = 1$.

The steady solutions of this problem have been computed using the first order backward Euler scheme with progressively higher time steps as the steady state is approached. Even for the highest Rayleigh number solutions converged to steady state in few iterations. Typical convergence histories of residuals on both grids using direct and iterative linear system solvers are displayed in Figure 2. Usually, higher-order solutions are started from the lower-order ones and this practice turns out very useful to reduce the overall computational cost of higher-order computations.

Figure 3 shows the temperature and the velocity components isolines and the qualitative improvement of the computed solutions increasing the degree of polynomial approximation.

A quantitative comparison of our most accurate solution ($80 \times 80$ grid, $P_3$ elements) with those reported by other authors can be found in Tables 1 and 2. The results used to compare the solutions are the Nusselt number (average, maximum and minimum and their locations) along the hot wall and the velocity components (maxima and their locations) along the centerlines. Following Le Quéré and Gjesdal maxima and minima have been found by evaluating the polynomial approximation at 1001 equidistant points and the velocity components have been scaled with $U = \Pr \sqrt{Ra} / (\nu L_r)$. The Nusselt number has
Figure 3: First benchmark problem ($Ra = 10^8$), isolines of $\theta$, $u$ and $v$, $40 \times 40$ grid.
\begin{center}
\begin{tabular}{|c|c|c|c|c|c|c|c|c|}
\hline
 & Present & Mayne\textsuperscript{20} & Gjesdal\textsuperscript{9} & Le Quéré\textsuperscript{18} & Present & Mayne\textsuperscript{20} & Gjesdal\textsuperscript{9} & Le Quéré\textsuperscript{18} \\
\hline
\hline
$Nu_{\text{max}}$ & 30.394 & 41.025 & 38.910 & 39.305 & $Nu_{\text{max}}$ & 87.224 & 91.209 & 86.700 & 87.236 \\
y & 0.018 & 0.039 & 0.020 & 0.018 & y & 0.008 & 0.007 & 0.009 & 0.008 \\
\hline
$Nu_{\text{min}}$ & 1.366 & 1.380 & 1.366 & 1.366 & $Nu_{\text{min}}$ & 1.919 & 2.044 & 1.919 & 1.919 \\
y & 1.0 & 1.0 & 1.0 & 1.0 & y & 1.0 & 1.0 & 1.0 & 1.0 \\
\hline
$u_{\text{max}} \cdot 10^2$ & 4.699 & 4.594 & 4.699 & 4.699 & $u_{\text{max}} \cdot 10^2$ & 3.219 & 2.831 & 3.219 & 3.219 \\
y & 0.879 & 0.884 & 0.879 & 0.879 & y & 0.928 & 0.946 & 0.928 & 0.928 \\

\vspace{1cm}
\hline
$v_{\text{max}} \cdot 10$ & 2.211 & 2.222 & 2.211 & 2.211 & $v_{\text{max}} \cdot 10$ & 2.222 & 2.223 & 2.222 & 2.222 \\
x & 0.021 & 0.021 & 0.021 & 0.021 & x & 0.012 & 0.013 & 0.012 & 0.012 \\
\hline
\end{tabular}
\end{center}

Table 1: First benchmark problem ($Ra = 10^7$), $80 \times 80$ grid, $P_3$ elements.

Table 2: First benchmark problem ($Ra = 10^8$), $80 \times 80$ grid, $P_3$ elements.

been evaluated as:

$$Nu = (\nabla_h \theta_h + r (\| \theta_h \|)) \big|_w \cdot n,$$

where the subscript $w$ means that the function inside parentheses is computed at the wall and the gradient of the temperature includes the contribution due to interface discontinuities. The data reported in Tables 1 and 2 indicate that our results are in excellent agreement with the best solutions available in the literature.

Table 3 summarizes the results of a convergence study for the $Ra = 10^8$ case. In this Table we have also reported inside parentheses the data computed with the artificial compressibility parameter $c = 0.1$. On the whole the results confirm the accuracy of the method here proposed. The influence of the artificial compressibility parameter is almost negligible for the high order approximation while larger changes can be observed for $P_1$ solutions. This is true in particular for the maximum of $u$ velocity component as can also be seen in Figure 4(a). Numerical experiments with $c = 10$ resulted in a poor or even impossible convergence of the solution.

4.2 Second Benchmark problem

This benchmark problem, proposed by Michalek et al.\textsuperscript{10}, is a slightly modified version of the previous test case. The square cavity is in this case filled with water and the temperatures of the isothermal walls, \textit{i.e.} $T_h = 10^0$ C and $T_c = 0^0$ C, are around the temperature of maximum density. The anomalous thermal variation of water density is described by the polynomial function

$$
\rho (T) = 999.840281167108 + 6.73268037314653 \cdot 10^{-2} T \\
- 8.94484552601798 \cdot 10^{-3} T^2 + 8.78462866500416 \cdot 10^{-5} T^3 \\
- 6.62139792627547 \cdot 10^{-7} T^4, \tag{18}
$$

which replaces Eq. (1) in the body force term of the momentum equation.
<table>
<thead>
<tr>
<th></th>
<th>$40 \times 40$</th>
<th>$40 \times 40$</th>
<th>$40 \times 40$</th>
<th>$80 \times 80$</th>
<th>$80 \times 80$</th>
<th>$80 \times 80$</th>
<th>Le Quéré $^{18}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\text{Nu}$</td>
<td>29.971</td>
<td>30.177</td>
<td>30.221</td>
<td>30.234</td>
<td>30.225</td>
<td>30.225</td>
<td>30.225</td>
</tr>
<tr>
<td>$y$</td>
<td>0.111</td>
<td>0.011</td>
<td>0.011</td>
<td>0.009</td>
<td>0.008</td>
<td>0.008</td>
<td>0.008</td>
</tr>
<tr>
<td>$\text{Nu}_{\text{min}}$</td>
<td>1.805</td>
<td>1.904</td>
<td>1.928</td>
<td>1.934</td>
<td>1.919</td>
<td>1.919</td>
<td>1.919</td>
</tr>
<tr>
<td>$y$</td>
<td>1.0</td>
<td>0.998</td>
<td>1.0</td>
<td>1.0</td>
<td>1.0</td>
<td>1.0</td>
<td>1.0</td>
</tr>
<tr>
<td>$u_{\text{max}} \cdot 10^2$</td>
<td>2.50646</td>
<td>3.14100</td>
<td>3.21221</td>
<td>2.72554</td>
<td>3.20760</td>
<td>3.21888</td>
<td>3.21875</td>
</tr>
<tr>
<td>$v_{\text{max}} \cdot 10$</td>
<td>(2.64600)</td>
<td>(3.18459)</td>
<td>(3.21558)</td>
<td>(3.30003)</td>
<td>(3.21979)</td>
<td>(3.21929)</td>
<td>(3.21929)</td>
</tr>
<tr>
<td>$x$</td>
<td>0.012</td>
<td>0.012</td>
<td>0.012</td>
<td>0.012</td>
<td>0.012</td>
<td>0.012</td>
<td>0.012</td>
</tr>
</tbody>
</table>

Table 3: First benchmark problem ($Ra = 10^8$), comparison of solutions with different polynomial approximation and grid resolution.

The Prandtl number is set to 13.31 and the Rayleigh number, at the reference temperature $T_r = 0^\circ C$, is equal to $1.503 \cdot 10^6$. It is known that the variation of viscosity is not negligible in the range of temperatures here considered, but for simplicity and with the aim of building an easily comparable benchmark solution all the fluid characteristics are assumed to be constant.

The computations have been performed on equally spaced, successively refined (from $4 \times 4$ to $128 \times 128$) quad grids, using the same time integration methods of the previous problem.

As suggested in Michalek $^{10}$, computed solutions have been compared on the basis of the average Nusselt number on the hot wall and of the maxima and minima of velocity components along centerlines. Values and positions of maxima and minima have been evaluated from the solution interpolated at 1001 equally spaced points. The results of a grid convergence study using $P_3$ elements are summarized in Table 4, where the velocities have been scaled with $U = Pr/(\nu L_r)$. For comparison purposes the last column in Table 4 reports the reference solution computed by Michalek et al. $^{10}$ on a $301 \times 301$ grid using the finite difference code FRECONV3V.

The non linear buoyancy term is responsible for the flow structure, shown in Figure 5,
with two counter-rotating vortices in the zone where the temperature reaches the maximum density value. One can also appreciate that this flow pattern is already captured on the $8 \times 8$ coarse grid. Finally, the velocity profiles of Figure 6 show that the $P_3$ solution is almost grid converged on the $16 \times 16$ grid.

4.3 Third benchmark problem

Unlike the previous test cases the last benchmark problem is unsteady. In this case the fluid is air ($Pr = 0.71$), the computational domain is an enclosed $8:1$ (height:width) rectangular cavity and the Rayleigh number, assuming the width as reference length $L_r$, is $3.4 \cdot 10^5$.

This problem was proposed at the MIT Conference on Computational Fluid and Solid Dynamics in 2001. Several contributors provided 32 sets of solution data using a broad range of computational methods. The results obtained by Le Quéré at al. using a Chebyshev collocation spectral method on $48 \times 180$ points were selected as the baseline for the comparison and the ranking of the methods. In a recent paper Gjesdal et al. reported the results of a spectral element simulation ($18^{th}$ order on a $4 \times 20$ grid) which fully agree with Le Quéré's data.

In this work the problem has been computed using $P_1$, $P_2$ and $P_3$ polynomial approximations and the grid resolutions suggested at the MIT Conference, namely $21 \times 101$ and $41 \times 201$. Points are clustered near the walls with $h_{min} = 0.018L_r$ and $h_{min} = 0.002L_r$ for the coarse and fine grids, respectively. Grid spacing increases in $x$ and $y$ directions with

Figure 4: First benchmark problem ($Ra = 10^8$), velocity components along centerlines.
Figure 5: Second benchmark problem, isolines of $\theta$, $u$ and $v$, $P_3$ elements.
(a) $u$ along $x = 0.5$  

(b) $v$ along $y = 0.5$  

Figure 6: Second benchmark problem, velocity components along centerlines, $P_3$ elements.

<table>
<thead>
<tr>
<th></th>
<th>4 × 4</th>
<th>8 × 8</th>
<th>16 × 16</th>
<th>32 × 32</th>
<th>64 × 64</th>
<th>128 × 128</th>
<th>T. Michalek$^{10}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$u_{\text{min}} \cdot 10^2$</td>
<td>-1.4209</td>
<td>-1.3420</td>
<td>-1.3123</td>
<td>-1.3098</td>
<td>-1.3098</td>
<td>-1.3098</td>
<td>-1.31</td>
</tr>
<tr>
<td>$x$</td>
<td>0.751</td>
<td>0.699</td>
<td>0.708</td>
<td>0.708</td>
<td>0.709</td>
<td>0.709</td>
<td>0.71</td>
</tr>
<tr>
<td>$u_{\text{max}}$</td>
<td>19.7566</td>
<td>4.3480</td>
<td>2.8749</td>
<td>2.8803</td>
<td>2.8003</td>
<td>2.8003</td>
<td>2.94</td>
</tr>
<tr>
<td>$x$</td>
<td>0.375</td>
<td>0.354</td>
<td>0.376</td>
<td>0.382</td>
<td>0.382</td>
<td>0.382</td>
<td>0.38</td>
</tr>
<tr>
<td>$v_{\text{min}} \cdot 10^2$</td>
<td>-1.3054</td>
<td>-1.7201</td>
<td>-1.7524</td>
<td>-1.7515</td>
<td>-1.7514</td>
<td>-1.7514</td>
<td>-1.75</td>
</tr>
<tr>
<td>$x$</td>
<td>0.705</td>
<td>0.691</td>
<td>0.705</td>
<td>0.705</td>
<td>0.705</td>
<td>0.705</td>
<td>0.70</td>
</tr>
<tr>
<td>$v_{\text{max}} \cdot 10^2$</td>
<td>1.7992</td>
<td>2.2130</td>
<td>2.1744</td>
<td>2.1712</td>
<td>2.1709</td>
<td>2.1709</td>
<td>2.17</td>
</tr>
<tr>
<td>$x$</td>
<td>0.064</td>
<td>0.041</td>
<td>0.041</td>
<td>0.042</td>
<td>0.042</td>
<td>0.042</td>
<td>0.04</td>
</tr>
<tr>
<td>$u_{\text{min}} \cdot 10^1$</td>
<td>-7.2125</td>
<td>-8.0671</td>
<td>-7.7815</td>
<td>-7.7826</td>
<td>-7.7827</td>
<td>-7.7827</td>
<td>-7.78</td>
</tr>
<tr>
<td>$y$</td>
<td>0.304</td>
<td>0.305</td>
<td>0.286</td>
<td>0.286</td>
<td>0.286</td>
<td>0.286</td>
<td>0.29</td>
</tr>
<tr>
<td>$u_{\text{max}} \cdot 10^1$</td>
<td>8.5708</td>
<td>8.5147</td>
<td>8.6746</td>
<td>8.6788</td>
<td>8.6789</td>
<td>8.6789</td>
<td>8.67</td>
</tr>
<tr>
<td>$y$</td>
<td>0.885</td>
<td>0.892</td>
<td>0.893</td>
<td>0.892</td>
<td>0.892</td>
<td>0.892</td>
<td>0.89</td>
</tr>
<tr>
<td>$v_{\text{min}} \cdot 10^1$</td>
<td>-11.0830</td>
<td>-10.333</td>
<td>-8.7888</td>
<td>-8.7872</td>
<td>-8.7856</td>
<td>-8.7856</td>
<td>-8.77</td>
</tr>
<tr>
<td>$y$</td>
<td>0.251</td>
<td>0.273</td>
<td>0.261</td>
<td>0.261</td>
<td>0.261</td>
<td>0.261</td>
<td>0.26</td>
</tr>
<tr>
<td>$y$</td>
<td>0.630</td>
<td>0.626</td>
<td>0.647</td>
<td>0.649</td>
<td>0.648</td>
<td>0.649</td>
<td>0.65</td>
</tr>
</tbody>
</table>

Table 4: Second benchmark problem, convergence study.
ratio equal to 1.2 and 1.05, respectively.

Since the problem is unsteady the solution has been advanced in time using the more accurate second-order Runge-Kutta time stepping scheme. In our fully implicit time discretization, the time step size is not subject to stability limitations and has been fixed on the basis of accuracy requirements. From numerical experiments we found that a time step equal to $\tau/40$, where $\tau = 3.4115L_r/U_r$ is the period of the solution computed by Le Quéré, was sufficiently small to assure accurate results. In fact, the maximum relative difference between two $P_2$ solutions on the $21 \times 101$ grid, computed doubling the time step size from $\tau/40$ to $\tau/20$, was less than 1%.

Also for this unsteady problem the $P_k$ solution was started from the $P_{k-1}$ approximation. We found that the $P_1$ approximation was not adequate to capture flow unsteadiness, even on the finest grid. Starting from the steady $P_1$ results, the $P_2$ and $P_3$ solutions have been advanced in time sequentially. For each approximation we found that about 15,000 time steps were largely sufficient to reach almost perfectly periodic solutions.

<table>
<thead>
<tr>
<th></th>
<th>$21 \times 101$</th>
<th>$21 \times 101$</th>
<th>$21 \times 101$</th>
<th>$41 \times 201$</th>
<th>$41 \times 201$</th>
<th>Le Quéré\textsuperscript{11}</th>
<th>Gjesdal\textsuperscript{9}</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\delta \cdot 10^4$</td>
<td>2.6546</td>
<td>2.6550</td>
<td>2.6548</td>
<td>2.6548</td>
<td>2.6547</td>
<td>2.6548</td>
<td>2.6561</td>
</tr>
<tr>
<td>$\theta' \cdot 10^2$</td>
<td>4.2443</td>
<td>4.2046</td>
<td>4.2872</td>
<td>4.2350</td>
<td>4.2885</td>
<td>4.2689</td>
<td>4.2774</td>
</tr>
<tr>
<td>$\varpi_1 \cdot 10^2$</td>
<td>5.6177</td>
<td>5.6175</td>
<td>5.6343</td>
<td>5.6262</td>
<td>5.6342</td>
<td>5.6342</td>
<td>5.6356</td>
</tr>
<tr>
<td>$u_1' \cdot 10^2$</td>
<td>5.4505</td>
<td>5.3903</td>
<td>5.5035</td>
<td>5.4323</td>
<td>5.5049</td>
<td>5.4767</td>
<td>5.4880</td>
</tr>
<tr>
<td>$p_{14} \cdot 10^3$\textsuperscript{(1)}</td>
<td>$-1.8618$</td>
<td>$-1.8508$</td>
<td>$-1.8492$</td>
<td>$-1.8654$</td>
<td>$-1.8515$</td>
<td>$-1.8536$</td>
<td>-</td>
</tr>
<tr>
<td>$p_{14}' \cdot 10^2$\textsuperscript{(1)}</td>
<td>2.0279</td>
<td>2.0087</td>
<td>2.0443</td>
<td>2.0226</td>
<td>2.0448</td>
<td>2.0355</td>
<td>-</td>
</tr>
<tr>
<td>$\varpi_1$</td>
<td>$-2.4030$</td>
<td>$-2.4012$</td>
<td>$-2.3695$</td>
<td>$-2.3637$</td>
<td>$-2.3721$</td>
<td>-</td>
<td>$-2.3678$</td>
</tr>
<tr>
<td>$\omega_1'$</td>
<td>1.0802</td>
<td>1.0643</td>
<td>1.0802</td>
<td>1.0755</td>
<td>1.0859</td>
<td>-</td>
<td>1.0801</td>
</tr>
<tr>
<td>$\overline{\delta}$</td>
<td>$-4.57939$</td>
<td>$-4.57940$</td>
<td>$-4.57946$</td>
<td>$-4.57944$</td>
<td>$-4.57946$</td>
<td>$-4.57946$</td>
<td>$-4.57933$</td>
</tr>
<tr>
<td>$\overline{\varpi}$</td>
<td>7.0750</td>
<td>7.0075</td>
<td>7.1241</td>
<td>7.0504</td>
<td>7.1241</td>
<td>7.0918</td>
<td>7.1026</td>
</tr>
<tr>
<td>$\overline{\varpi}$</td>
<td>2.3949</td>
<td>2.3948</td>
<td>2.3951</td>
<td>2.3950</td>
<td>2.3951</td>
<td>-</td>
<td>2.395</td>
</tr>
<tr>
<td>$\overline{u}' \cdot 10^5$</td>
<td>3.4021</td>
<td>3.3366</td>
<td>3.3742</td>
<td>3.3444</td>
<td>3.3770</td>
<td>-</td>
<td>3.354</td>
</tr>
<tr>
<td>$\overline{\varpi}$</td>
<td>3.0166</td>
<td>3.0165</td>
<td>3.0171</td>
<td>3.0177</td>
<td>3.0171</td>
<td>-</td>
<td>3.0171</td>
</tr>
<tr>
<td>$\overline{\varpi}' \cdot 10^5$</td>
<td>3.1964</td>
<td>3.1612</td>
<td>3.2078</td>
<td>3.1773</td>
<td>3.2090</td>
<td>-</td>
<td>3.198</td>
</tr>
</tbody>
</table>

Table 5: Third benchmark problem, convergence study.

Table 5 summarizes our computational results. All mean data reported in the Table have been averaged over 20 periods. The monitored quantities are the averaged value and the peak to peak amplitude of the variables at the first control point (prescribed at the $p_{14}$ is the pressure difference between the first and the fourth control point ($x_4 = 0.819L_r$ and $y_4 = 7.37L_r$).
MIT conference as \( x_1 = 0.181L_r \) and \( y_1 = 7.37L_r \), the averaged Nusselt number along the hot wall and the averaged velocity magnitude \((\bar{u})\) and vorticity \((\bar{\omega})\) over the whole computational domain. Our results are in excellent agreement with the reference data and we mention that, using the overall metrics of the errors defined at the MIT conference for the comparison of the methods, both the coarse and the fine grid \( P_3 \) solutions would have been ranked first, \textit{i.e.} closest to the reference solution, and the \( P_2 \) solutions sixth and second.

All the solutions of this test case have been computed running the code in parallel and using the block Jacobi preconditioned GMRES algorithm of the PETSc library for the linear algebra. A relative residual error of \( 10^{-8} \) was used as stopping criterion for the iterative solver. Typically, the \( P_3 \) computations on the coarse grid took 120 GMRES iterations to converge the solution within the prescribed tolerance at each time step.

![Figure 7: Third benchmark problem, isolines of the instantaneous temperature deviations from the local time averages (21 x 101 grid, \( P_3 \) elements, pictures 1/5 of the period apart, contour interval \( \Delta \theta = 0.002 \), dashed negative values, \( \theta = 0 \) contour omitted for clarity).](image-url)
5 CONCLUSIONS

In this work a novel high-order DG method for the numerical solution of the incompressible Navier-Stokes equations has been extended to deal with natural convection problems.

The idea of using the solution of perturbed 1D Riemann problems for the purpose of inter-element flux computation has been successfully applied to the discretization of the convective term in the energy conservation equation.

The method has been validated by computing a number of well known steady and unsteady natural convection benchmark problems. In all cases our DG solutions were in excellent agreement with the best reference results for such problems.

REFERENCES


