A SECOND ORDER SPLITTING ALGORITHM FOR THERMALLY-DRIVEN FLOW PROBLEMS

P. D. MINEV, F. N. VAN DE VOSSE, L. J. P. TIMMERMANS AND A. A. VAN STEENHOVEN

Department of Mechanical Engineering, Eindhoven University of Technology, P.O. Box 513, 5600 MB Eindhoven, The Netherlands

ABSTRACT
A splitting technique for solutions of the Navier-Stokes and the energy equations, in Boussinesq approximation, is presented. The equations are first integrated in time using a splitting procedure and then discretized spatially by means of a high-order spectral element method. The whole technique is validated on the flow in a differentially-heated cavity at intermediate and transitional Rayleigh numbers. The results are in a very good agreement with other available numerical solutions.

KEY WORDS Operator splitting Spectral element method Differentially-heated cavity Thermally-driven flows

INTRODUCTION
Highly convection-dominated thermally-driven problems appear frequently in engineering practice. Their investigation often involves the necessity of direct simulation of transition to turbulence, which requires high accuracy of spatial and time discretization. In the present study a numerical technique for simulations of thermally-driven flows is presented. The spatial discretization is based on the spectral element method (SEM) proposed by Patera. The SEM is a high-order Galerkin method that demonstrates excellent properties (small numerical diffusion and dispersion error) with respect to convection-dominated problems (see Timmermans and van de Vosse). It combines some of the ideas of the spectral methods (see Canuto et al.) and the domain decomposition, typical of the finite element methods. Thus, it allows increasing accuracy of the approximation by both increasing the number of the elements and the order of the polynomial approximation. If the second way is used an exponential decrease of the error can be expected in the case of sufficiently regular (incompressible) flows (see Maday and Patera). A major problem related to such high-order methods is their efficiency, because the resulting matrix is quite full in comparison to the low-order methods. That is why their application to fluid flow problems is often combined with some splitting procedures, allowing a decomposition of the Navier-Stokes and energy equations into a set of positive definite symmetric algebraic systems that can be efficiently solved by means of iterative methods (see for example Karniadakis et al.). An extensive analysis of the theory of the splitting techniques for the incompressible flow equations can be found in Gresko.

The splitting scheme presented in this paper treats the convection part explicitly with a third order Runge-Kutta scheme. The diffusion part and the pressure/incompressibility constraint are treated implicitly, but also separately by means of a pressure-correction scheme presented in Timmermans et al. Its main advantage is that the resulting velocity satisfies the discrete incompressibility constraint and is also correctly diffused contrary to some other splitting schemes.
which either violate the first or the second of these properties (see Orszag et al.\textsuperscript{8}). The decoupling of the temperature and velocity is realized by using an explicit extrapolation for the velocity during the convection of the temperature and then using the diffused temperature on the current time level during the diffusion step of the Navier-Stokes equations. It should be noted that although the splitting technique presented below is used in combination with the SEM, it can also be used with other types of spatial discretization because it only concerns the time integration of the equations and finally requires solution of only three Helmholtz (two for the velocity components and one for the temperature) and one Poisson equation (for the pressure correction) each time-step.

The method presented in this study is validated on the flow in a differentially-heated square cavity that is frequently used to evaluate the performance of the numerical techniques (see de Vahl Davis and Jones\textsuperscript{9}). A benchmark solution is provided by de Vahl Davis\textsuperscript{10}. It concerns the flow at Prandtl number $Pr = 0.71$ (air) and Rayleigh number between $10^3$ and $10^6$. Furthermore, results of the direct numerical simulation of the flow in transitional regime at $R = O(10^6)$ are presented. These results are discussed in the light of other available numerical results (see Paolucci and Chenowith\textsuperscript{11}).

**MATHEMATICAL MODEL**

In order to model many non-isothermal flows of practical interest, it is usually sufficient to assume that the density and viscosity of the flow are all temperature independent except for the density in the source term of the momentum equations, which results in the so-called Boussinesq equations,

\[
\frac{\partial u}{\partial t} + (u \cdot \nabla)u = -\nabla p + R \cdot Pr \cdot T g + Pr \nabla^2 u
\]

\(\nabla \cdot u = 0\) \hspace{2cm} (1)

\[
\frac{\partial T}{\partial t} + (u \cdot \nabla)T = \nabla^2 T
\]

where $R = (g \beta \Delta T l^3)/(\nu \kappa)$, $Pr = \nu/\kappa$ are the commonly used Rayleigh and Prandtl numbers and $g = (0, 1)^T$. Here, $g$ is the acceleration of gravity, $\beta$ is the thermal expansion coefficient, $l$ is the characteristic length, $\Delta T$ is the characteristic temperature difference, $\kappa$ is the thermal diffusivity and $\nu$ is the kinematic viscosity of the fluid. The equations are non-dimensionalized with respect to the following characteristic velocity and pressure: $U = \kappa/l$, $P = \rho \kappa^2/l^2$. This scaling is discussed in detail by Paolucci and Chenowith\textsuperscript{11}.

**NUMERICAL ALGORITHM**

There are three main problems related to the numerical solution of the system (1)-(3). The first concerns the treatment of the convection operators because of their non-linearity and asymmetry. The second problem is related to the coupling between the velocity and temperature via the source term in the momentum equation and the convection part of the energy equation. The third is how to impose the incompressibility constraint and how to calculate the pressure.

Since the convection operator is more suitable for an explicit treatment in the present technique, a so-called operator-splitting approach is adopted (see Timmermans et al.\textsuperscript{12}). It is discussed in detail by Maday et al.\textsuperscript{13}. If the convection part of (1)-(3) is split in such a manner and the resulting diffusion problem is discretized in time with a second-order-backward-difference scheme,
the following semi-discrete system appears,

\[
\frac{3u^{n+1} - 4u^n + 2u^{n-1}}{2\Delta t} = -\nabla p^{n+1} + Pr\nabla^2 u^{n+1} + R \cdot PrT^{n+1} + g
\]  

(4)

\[
\nabla \cdot u^{n+1} = 0
\]  

(5)

\[
\frac{3T^{n+1} - 4\bar{T}^n + \bar{T}^{n-1}}{2\Delta t} = \nabla^2 T^{n+1}
\]  

(6)

Here, the quantities marked by \(\sim\) are the corresponding quantities at level \(n-i\) \((i=0,1)\), convected according to the following equation,

\[
\frac{\partial \bar{Q}^{n-i}(s)}{\partial s} = (u(s) \cdot \nabla) \bar{Q}^{n-i}, \quad 0 \leq s \leq (i + 1)\Delta t, \quad i = 0, 1
\]  

\[
\bar{Q}^{n-i} = Q^{n-i}
\]  

(7)

where \(Q\) is either \(u\) or \(T\).

As the eigenvalues of the discrete spectral element convection operator are distributed in the vicinity of the imaginary axis problem (7) can be best solved by means of a multistep explicit scheme. Taking the stability regions of the multistep schemes into account a 3-step Runge-Kutta scheme is chosen. For the non-linear velocity problem is reads,

\[
\bar{u}^{n-i}_{m+1/3} = \bar{u}^{n-i}_m - \Delta s \left( \frac{1}{3} (\bar{u}^{n-i}_{m} \cdot \nabla) \bar{u}^{n-i}_m \right)
\]  

\[
\bar{u}^{n-i}_{m+1/2} = \bar{u}^{n-i}_m - \Delta s \left( \frac{1}{2} (\bar{u}^{n-i}_{m+1/3} \cdot \nabla) \bar{u}^{n-i}_m + \bar{u}^{n-i}_{m+1/3} \right)
\]  

\[
\bar{u}^{n-i}_{m+1} = \bar{u}^{n-i}_m - \Delta s (\bar{u}^{n-i}_{m+1/2} \cdot \nabla) \bar{u}^{n-i}_m + \bar{u}^{n-i}_{m+1/2}
\]  

\[
m = 0, \ldots, M, \quad M = \frac{\Delta t}{\Delta s}
\]  

\[
\bar{u}^{n-i}_0 = u^{n-i}, \quad i = 0, 1
\]  

(8)

Here, \(\Delta s\) is the “convection” time step that can be chosen smaller than the “diffusion” time step \(\Delta t\).

The stability region of this scheme is not too restrictive for the convection operator, because in the case of a spectral discretization its eigenvalues have a modulus of order \(O(N^2)\) with \(N\)—the order of the elements used. This is not the case with the discrete Laplacian that has eigenvalues in the order \(O(N^4)\) (see Timmermans\(^{14}\)). The convection of the temperature is more complicated because then the velocity on the corresponding time level is necessary. In order to maintain the second order of accuracy of the whole scheme and to decouple the velocity and temperature, a second-order approximation has to be used for the velocity at the moments \(t^n + \tau\) \((\tau = (m+\theta)\Delta s, \ m = 0, \ M - 1; \ \theta = 0, 1/3, 1/2, 1)\). In general two options are available. The first one is to “diffuse” the velocity according to (4)-(5) with a source term \(R \cdot Pr \cdot T^n\) and then to interpolate its value for \(t^n < t < t^{n+1}\). For many flows of practical interest, however, this term is dominant in the momentum equations because the Rayleigh number is very high. That is why the second option is adopted here: first, convect the velocity with an explicit second order extrapolation for the velocity at \(t = t^n + \tau\),

\[
u^{n+\tau} = (1 + \tau/\Delta t)u^n - \tau/\Delta t u^{n-1}
\]  

(9)

and then “diffuse” it according to (6). Then the velocity/pressure saddle-point problem (4)-(5) can be solved with an implicit source term. There are several ways of doing that but in the context of the splitting of the convection part and the energy equation the most attractive way
is to use some continuous projection procedure. The word "continuous" is used in the sense that it is performed only in terms of time discretization without assuming any spatial discretization of the equations. Such a procedure is discussed in Timmermans et al. and includes the following steps:

1. First an intermediate velocity field \( u^* \) is calculated from (4) by choosing the pressure at the previous time level,

\[
\frac{3u^* - 4u^n + u^{n-1}}{2\Delta t} = -\nabla p^n + Pr\nabla^2 u^* + R \cdot Pr T^{n+1} \quad (10)
\]

2. An equation for the pressure correction \( p^c = p^{n+1} - p^n \) is obtained by subtracting (10) from the original equation (4),

\[
\frac{3}{2}(u^{n+1} - u^*) - \Delta t Pr \nabla^2 (u^{n+1} - u^*) = -\Delta t \nabla p^c \quad (11)
\]

Further, a Poisson equation for the pressure correction can be derived by applying the divergence operator to both sides of (11) and neglecting the second term on its left-hand side (see Hawken et al.\textsuperscript{15}). It can be shown, however, that the resulting solution \((u, p)\) does not satisfy the non-split semi-discrete system (4)-(5) because of the latter neglection. This worsens the solution considerably if high-order methods are used for spatial discretization (see Timmermans et al.\textsuperscript{7}). If this term is not neglected and (5) is imposed on \( u^{n+1} \) (assuming sufficient regularity of \( u \) and \( p \)) the following equation appears,

\[
\nabla^2 q = \frac{3\nabla \cdot u^*}{2\Delta t} \quad (12)
\]

which is again a Poisson equation but for another quantity: \( q = p^c + Pr \nabla \cdot u^* \).

3. Now the velocity and pressure time level \( n+1 \) can be calculated according to:

\[
u^{n+1} = u^* - \frac{2}{3\Delta t} \nabla q \quad (13)
\]

\[
p^{n+1} = p^n + q - Pr \nabla \cdot u^* \quad (14)
\]

As it is proved by Timmermans et al.\textsuperscript{7} velocity and pressure satisfy the original saddle-point problem. The details concerning the boundary conditions for the equations above can be found in Timmermans et al.\textsuperscript{7}. The procedure described above decomposes the Stokes problem (4)-(5) into two Helmholtz equations for the intermediate velocity components and a Poisson equation for the pressure correction.

As mentioned above, a spectral element discretization is applied to the semi-discrete systems (8) and (10)-(14). The SEM is a high-order Galerkin technique that uses the Gauss-Lobatto points as collocation and integration points (resulting in a diagonal mass matrix). Similar to the classical finite element methods, it also allows decomposition of the physical domain into isoparametric quadrilaterals. More details can be found in Maday and Patera\textsuperscript{4} and Timmermans\textsuperscript{14}.

**RESULTS**

The modified pressure-correction scheme has been extensively tested and validated for isothermal flows by Timmermans et al.\textsuperscript{7}. Here, the algorithm described above is validated on the benchmark solution provided by de Vahl Davis\textsuperscript{16}. It concerns the flow in a differentially-heated square cavity at \( Pr=0.71 \) and 4 different values of the Rayleigh number: \( R = 10^3, 10^4, 10^5, 10^6 \). The geometry of the problem and the boundary conditions are presented in Figure 1. This problem has been extensively investigated by Markatos and Pericleous\textsuperscript{16} and Lankhorst\textsuperscript{17} under the same conditions.

In the present calculations the steady state (if it exists) is reached through an unsteady integration in time using the algorithm described above. The initial conditions are chosen to be
zero velocity, pressure and temperature. Then the temperature on the left-hand side wall is switched to 1 and the unsteady algorithm described above is applied. After some initial transience the flow reaches a steady state for Rayleigh numbers up to $10^7$.

The streamlines and isotherms of the flow at different Rayleigh numbers are presented in Figures 2–5. They are in very good agreement with all the available numerical and experimental data (see Lankhorst and Markatos and Pericleous). The results at $R = 10^3, 10^4, 10^5, 10^6$ for the maximal horizontal and vertical velocity components at $x=0.5$ and $y=0.5$, respectively, and the minimal and maximal value of the Nusselt number on the “hot” wall are compared with the benchmark solution of de Vahl Davis in Table I. The Nusselt number is given by: $Nu = \partial T/\partial x|_{x=0}$. Its values in Table I are computed by means of a simple first order finite-difference

![Figure 1](image1) Buoyancy-driven flow in an enclosed cavity. Computational domain with boundary conditions

![Figure 2](image2) Streamlines (left) and isotherms (right) for the buoyancy-driven flow in an enclosed cavity at $R = 10^3$. Modified pressure correction/operator splitting scheme using $4 \times 4$ elements of degree $N=8$

![Figure 3](image3) Streamlines (left) and isotherms (right) for the buoyancy-driven flow in an enclosed cavity at $R = 10^4$. Modified pressure correction/operator splitting scheme using $4 \times 4$ elements of degree $N=8$
Table 1  Buoyancy-driven flow in an enclosed cavity. Present results (P) compared with the benchmark numerical solution (B) and the derivation (D) for $R=10^3$ through $R=10^6$. Modified pressure correction/operator splitting scheme using $4 \times 4$ elements of degree $N=8$.

<table>
<thead>
<tr>
<th>Variable</th>
<th>Source</th>
<th>$R=10^3$</th>
<th>$R=10^4$</th>
<th>$R=10^5$</th>
<th>$R=10^6$</th>
<th>$R=10^6$*</th>
</tr>
</thead>
<tbody>
<tr>
<td>$u_{1, \text{max}}$</td>
<td>B</td>
<td>3.649</td>
<td>16.178</td>
<td>34.73</td>
<td>64.63</td>
<td>64.63</td>
</tr>
<tr>
<td></td>
<td>P</td>
<td>3.630</td>
<td>16.171</td>
<td>34.15</td>
<td>63.02</td>
<td>68.17</td>
</tr>
<tr>
<td></td>
<td>D (%)</td>
<td>-0.5</td>
<td>0.0</td>
<td>-1.6</td>
<td>-2.3</td>
<td>+5.5</td>
</tr>
<tr>
<td>$x_3(u_1)$</td>
<td>B</td>
<td>0.813</td>
<td>0.823</td>
<td>0.855</td>
<td>0.850</td>
<td>0.850</td>
</tr>
<tr>
<td></td>
<td>P</td>
<td>0.830</td>
<td>0.830</td>
<td>0.875</td>
<td>0.830</td>
<td>0.844</td>
</tr>
<tr>
<td>$u_{2, \text{max}}$</td>
<td>B</td>
<td>3.697</td>
<td>19.617</td>
<td>68.59</td>
<td>219.39</td>
<td>219.39</td>
</tr>
<tr>
<td></td>
<td>P</td>
<td>3.693</td>
<td>19.604</td>
<td>66.85</td>
<td>219.69</td>
<td>219.98</td>
</tr>
<tr>
<td></td>
<td>D (%)</td>
<td>-0.1</td>
<td>-0.1</td>
<td>-2.5</td>
<td>+0.1</td>
<td>-3.3</td>
</tr>
<tr>
<td>$x_3(u_2)$</td>
<td>B</td>
<td>0.178</td>
<td>0.119</td>
<td>0.066</td>
<td>0.0379</td>
<td>0.0379</td>
</tr>
<tr>
<td></td>
<td>P</td>
<td>0.170</td>
<td>0.125</td>
<td>0.079</td>
<td>0.0404</td>
<td>0.0313</td>
</tr>
<tr>
<td>$N_{\text{u max}}$</td>
<td>B</td>
<td>1.505</td>
<td>3.528</td>
<td>7.717</td>
<td>17.925</td>
<td>17.925</td>
</tr>
<tr>
<td></td>
<td>D (%)</td>
<td>+0.1</td>
<td>+0.1</td>
<td>0.0</td>
<td>-3.2</td>
<td>-20.95</td>
</tr>
<tr>
<td>$x_3(N_{\text{u}})$</td>
<td>B</td>
<td>0.092</td>
<td>0.143</td>
<td>0.081</td>
<td>0.0378</td>
<td>0.0378</td>
</tr>
<tr>
<td></td>
<td>P</td>
<td>0.080</td>
<td>0.125</td>
<td>0.080</td>
<td>0.0404</td>
<td>0.0625</td>
</tr>
<tr>
<td>$N_{\text{u min}}$</td>
<td>B</td>
<td>0.692</td>
<td>0.586</td>
<td>0.729</td>
<td>0.989</td>
<td>0.989</td>
</tr>
<tr>
<td></td>
<td>P</td>
<td>0.692</td>
<td>0.586</td>
<td>0.726</td>
<td>0.972</td>
<td>0.989</td>
</tr>
<tr>
<td></td>
<td>D (%)</td>
<td>0.0</td>
<td>0.0</td>
<td>-0.3</td>
<td>-1.7</td>
<td>0.0</td>
</tr>
<tr>
<td>$x_3(N_{\text{u}})$</td>
<td>B</td>
<td>1.0</td>
<td>1.0</td>
<td>1.0</td>
<td>1.0</td>
<td>1.0</td>
</tr>
<tr>
<td></td>
<td>P</td>
<td>1.0</td>
<td>1.0</td>
<td>1.0</td>
<td>1.0</td>
<td>1.0</td>
</tr>
</tbody>
</table>

*These results were obtained using $16 \times 16$ finite elements of degree $N=2$. 

Figure 4  Streamlines (left) and isotherms (right) for the buoyancy-driven flow in an enclosed cavity at $R=10^3$. Modified pressure correction/operator splitting scheme using $4 \times 4$ elements of degree $N=8$.

Figure 5  Streamlines (left) and isotherms (right) for the buoyancy-driven flow in an enclosed cavity at $R=10^6$. Modified pressure correction/operator splitting scheme using $4 \times 4$ elements of degree $N=8$. 

* These results were obtained using $16 \times 16$ finite elements of degree $N=2$. 

© Emerald Backfiles 2007
formula. The mesh consists of $4 \times 4$ equal spectral elements of order 8 (33 $\times$ 33 grid points). The "diffusion" time step is adjusted experimentally and it varies a lot with the increase of the Rayleigh number. At $R = 10^3$ it is equal to 0.004 and at $R = 10^6$ its value is 0.00004. The convection step $\Delta s$ is adjusted according to the CFL condition.

The results are in very good agreement with the extrapolated data of de Vahl Davis\textsuperscript{10} for mesh size tending towards 0. The maximal difference is observed for the values of the Nusselt number at $R = 10^6$. However, its value is very sensitive to the position of the point where it is calculated (see also Lankhorst\textsuperscript{17}). The observed and relatively large deviations of the velocity components at $R = 10^5$, $10^6$ are due to the relatively large differences of the coordinates of the points where they are measured. In all the cases (also for the Nusselt number) the corresponding values in our calculations are measured in the points of our mesh that are in closest position to the points given by de Vahl Davis\textsuperscript{10}. However, it is practically impossible to use the same mesh as in the last work because of the non-equidistant distribution of the Gauss-Lobatto points used in the present calculations. In order to check the performance of the high-order approximation compared to the low-order methods independently of the time discretization, the calculations at $R = 10^6$ are carried out using the splitting procedure described above with $16 \times 16$ elements of order 2 (finite elements). This mesh contains the same number of points as the mesh used in the SEM simulation. The resulting values of the same parameters as given by de Vahl Davis\textsuperscript{10} are presented in the last column of Table 1. They clearly differ from the benchmark solution much more than the SEM results. The CPU-time per one step (on a SG Challenge computer) is 4.33 s for the finite element simulation and 7.57 s for the high-order elements. In order to dampen the initial transience 0.32 dimensionless time is necessary in the first case and 0.26 in the second. Thus, the CPU-time for the high-order calculations is not much longer but the results are essentially better. Except for the superior accuracy properties of the SEM in general, another possible reason is that in both cases a diagonal mass matrix is used, which is not a serious problem for the high-order methods but can cause additional loss of accuracy with low-order approximation (see Timmermans and van de Vosse\textsuperscript{2}).

In order to examine the behaviour of the method at transitional regimes of the flow some experiments at higher Rayleigh number values were carried out. At $R = 10^7$ the flow remains essentially steady (after some initial disturbance is damped) but a mesh refinement is necessary. At $R = 10^8$ it is practically impossible to start with zero initial data and therefore the results at $R = 10^7$ are used as initial conditions. At this value of the Rayleigh number the numerical technique demonstrates some weak instability which is clearly artificial. On a mesh of $6 \times 6$ equal elements of order 8 an essentially unsteady flow is obtained. When the mesh is refined near the walls employing $8 \times 8$ elements of order 8, the flow clearly dampens all the oscillations and monotonically tends to a steady state (see Figure 6). The same feature of the high-order methods

\begin{figure}[h]
\centering
\includegraphics[width=0.5\textwidth]{figure6}
\caption{Isotherms for the buoyancy-driven flow in an enclosed cavity at $R = 10^8$. Mesh of $6 \times 6$ equal elements of degree $N = 8$ (left) and $8 \times 8$ elements of degree $N = 8$ refined at the walls (right)}
\end{figure}
is mentioned by Gresho et al. The reason for this behaviour in the present case seems to be the insufficient resolution of the very thin boundary layer on the vertical walls because the unsteadiness is clearly initiated here. Note that this spurious oscillatory solution (only accidentally) resembles Tollmien-Schlichting waves typical of a boundary-layer transition.

Paolucci and Chenoweth indicate that the flow clearly exhibits unsteady behaviour at $R=2 \times 10^8$ and it dampens all the initial oscillations at $R=1.9 \times 10^8$. The frequency of the oscillations is found to be equal to 630.3. The present simulations indicate the same behaviour of the flow at $R=2 \times 10^8$. The power spectrum of the second velocity component at the point $(0.1029, 0.80676)$ (the point of our mesh that is closest to the sample point of Paolucci and Chenowith) is presented in Figure 7. The frequency peak appearing there is at about 604, which is 4.1% lower than in the results cited above.

---

![Figure 7](image_url)  
*Figure 7* Time history (top) and power spectrum (bottom) of second component of the velocity in the point $(0.1029, 0.80676)$ at $R=2 \times 10^8$
CONCLUSIONS

The application of high-order spectral element methods to incompressible flows and heat transfer equations is quite efficient, if it is combined with a high-order splitting procedure for time integration. The present technique suggests a way to do that. It performs an operator splitting of the convection part of the spatially continuous equations first and then treats the resulting diffusion part of the Navier-Stokes equations with a projection procedure again without assuming any spatial discretization. The convection of both velocity and temperature is performed with an efficient explicit scheme allowing to decouple them completely. The temperature is convected using a second-order extrapolation for the velocity, which seems to be better for convection-dominated flows than the use of an explicit source term in the Navier-Stokes equations. The solution of the whole system then requires solution of three Helmholtz equations for the velocity and temperature and one Poisson equation for the pressure correction. Thus, it allows for the use of a non-staggered grid for the velocity and pressure. There is a variety of iterative methods that can solve these equations quite effectively (for example the finite-element preconditioning presented by Timmermans and van de Vosse\(^\text{19}\)). Since the convection is treated explicitly, in some cases (especially when unstable flow regimes are simulated) it can cause severe stability restrictions on the time step. Instead of the three-step Runge-Kutta scheme used at present, one can use a four-step scheme or a four-level Adams-Bashforth explicit scheme for such a case.

The procedure is validated on the flow in a differentially-heated cavity that is commonly used for such purposes. The results at intermediate values of the Rayleigh number are in very good agreement with an available benchmark solution. Moreover, the technique performs quite well at transitional values of this parameter, predicting the same critical value as in some other numerical studies and a frequency of the oscillations close to the data available in the literature. However, the spectral element mesh has to be carefully chosen, especially when oscillatory regime is simulated, because the results discussed above show that insufficient resolution can cause spurious oscillations in the flow.

ACKNOWLEDGEMENT

The first author was financially supported by the J. M. Burgers Centre for Fluid Mechanics during the work described in this paper. The third author was supported by the Dutch Technology Foundation (STW) grant No EWT 88.1442.

REFERENCES