Segmentation based Boundary Domain Integral Method for the numerical solution of Navier–Stokes equations

M. Hriberšek
Faculty of Mechanical Engineering, Institute of Power, Process and Environmental Engineering, University of Maribor, Slovenia

Abstract

This contribution deals with further development of Boundary Domain Integral algorithm for computation of laminar viscous fluid flows governed by the Navier-Stokes equations. The algorithm uses the velocity-vorticity formulation and is based on vector-potential formulation of flow kinematics. This results in an accurate determination of the boundary vorticity values, a crucial step in constructing an accurate numerical algorithm for the computation of flows in complex geometries, i.e. geometries with sharp corners. In order to lower computational costs the domain velocity computations are done by the segmentation technique using large subdomains. After the kinematics equation is resolved, the vorticity transport equation is solved using a macro-element approach. This enables us to use a macro-element based diffusion-convection fundamental solution, a key factor in assuring accuracy of the computations for high Reynolds number flows. The proposed numerical algorithm is tested on several test problems, including the standard driven cavity and backward facing step flow, together with driven cavity flow in an L shaped cavity. The comparison of computational results show that the developed algorithm is capable of an accurate resolution of the flow fields in complex geometries.

1 Introduction

In the context of BEM related methods for viscous fluid flows several successful attempts have already been made, see [3], [5] and [6] among others and an excel-
lent survey of these approaches can be found in [4]. These numerical approaches were based on different forms of the Navier-Stokes equations, representing the frame for the solution of viscous flow problems. They also developed different techniques for capturing nonlinear domain effects, including internal cells, macro-elements and dual and multiple reciprocity methods.

Different approaches were successful also in cases of higher values of Re number flows, although the highest values, as for the case of driven cavity, computed with other established approximation methods, is Re=10,000, were not easily obtained, [1] (Re=10,000), or were restricted to lower or moderately high Re number values, [6] (up to Re=1000) and [5] (up to Re=5000).

The present contribution presents a novel BDIM technique, formally equal to the vector potential formulation with segmentation of flow kinematics together with macro-element based computation of flow kinetics with the use of diffusion-convection fundamental solution. The result is a stable numerical technique which is able to accurately predict high value Re flows in complex geometries.

2 Governing equations

Since we are dealing with incompressible fluid flow in planar geometry, the set of governing equations are as follows:

- continuity equation:

\[
\frac{\partial v_i}{\partial x_i} = 0,
\]

(1)

- momentum transport equation:

\[
\frac{\partial v_i}{\partial t} + v_j \frac{\partial v_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial P}{\partial x_i} + \nu \frac{\partial^2 v_i}{\partial x_j \partial x_j}.
\]

(2)

The present investigation deals with constant material properties, i.e. mass density, \(\rho\), and kinematic viscosity, \(\nu\).

For the numerical solution of the Navier-Stokes equations (1) and (2), we choose the velocity-vorticity formulation. When using this formulation, the dynamics of a viscous incompressible fluid is partitioned into its kinematic and kinetic aspect, i.e.

- flow kinematics:

\[
\frac{\partial^2 v_i}{\partial x_j \partial x_j} + \epsilon_{ij} \frac{\partial \omega}{\partial x_j} = 0,
\]

(3)

- flow kinetics:

\[
\frac{\partial \omega}{\partial t} + v_j \frac{\partial \omega}{\partial x_j} = \nu \frac{\partial^2 \omega}{\partial x_j \partial x_j}.
\]

(4)

3 Integral representations

The unique advantage of the Boundary Domain Integral Method originates from the application of Green’s fundamental solutions as particular weighting functions.
Since they only consider the linear transport phenomenon, an appropriate selection of linear differential operators is of major importance in establishing stable and accurate singular integral representations corresponding to original differential conservation equations.

With the use of Green’s theorem and the selection of the Laplace fundamental solution the integral statement of flow kinematics is derived ([2]) as follows:

\[
c(\xi) v_i(\xi) + \int_{\Gamma} v_i \frac{\partial u^*}{\partial n} d\Gamma = \int_{\Gamma} v_j \frac{\partial u^*}{\partial t} d\Gamma - e_{ij} \int_{\Omega} \omega \frac{\partial u^*}{\partial x_j} d\Omega. \tag{5}
\]

In the case of the flow kinetics, the use of Green’s identities and diffusion-convection fundamental solution leads to the integral equation [3]:

\[
c(\xi) \omega(\xi) + \int_{\Gamma} \omega \frac{\partial u^*}{\partial n} d\Gamma = \frac{1}{\nu} \int_{\Gamma} (\nu \frac{\partial \omega}{\partial n} - \omega v_n) u^* d\Gamma + \frac{1}{\nu} \int_{\Omega} \omega \hat{v}_j \frac{\partial u^*}{\partial x_j} d\Omega + \frac{1}{\nu \Delta t} \int_{\Omega} \omega_{F-1} u^* d\Omega, \tag{6}
\]

with \( v_n = v_j n_j \) and decomposition of the velocity field \( v_j \) into an average constant vector \( \bar{v}_j \) and a perturbated one \( \hat{v}_j \), i.e. \( v_j = \bar{v}_j + \hat{v}_j \).

4 Subdomain and macro element technique

The set of integral forms for the governing equations (5) and (6) is transformed to its algebraic form by dividing the boundary \( \Gamma \) into boundary elements \( \Gamma_e \) and the domain \( \Omega \) into domain cells \( \Omega_c \) and by approximating the field functions by quadratic interpolation functions.

In this contribution a novel numerical approach to the BDIM, which combines the segmentation technique for the flow kinematics, [2], with the macro-element approach for the flow kinetics, i.e. vorticity transport, [3], is developed. In this way the algorithm combines conservation properties of the vector-potential formulation of the flow kinematics and accurate computation of diffusion-convection transport phenomena, performed by a macro-element based computation of the flow kinetics.

![Figure 1: Sequence of computational meshes.](image-url)
The new numerical approach is composed of the following steps:

1. Computation of the boundary vorticity and velocity values from flow kinematics equation, discretized by the continuous full domain discretization, Figure 1(a).
2. Computation of the velocity values on the subdomain interfaces.
3. Computation of the velocity values inside the subdomains, Figure 1(b).
4. Transformation of the velocity and vorticity values from the subdomain mesh (Ωᵢ) to the macro element mesh (Ωₘₑ).
5. Computation of the domain vorticity values and the vorticity fluxes from the macro element mesh.
6. Transformation of vorticity values from the macro element mesh (Ωₘₑ) to the subdomain mesh (Ωᵢ).

5 Test problems

In order to show the accuracy and stability of the proposed numerical algorithm, first the standard driven cavity problem and then two test cases with nonregular geometries have been investigated. The presence of nonregular boundaries significantly affect the fluid flow conditions as compared with flows in confined square cavities and this presents a challenge for all CFD numerical algorithms.

5.1 Driven cavity flow

Fluid flow in a square cavity is one of the most difficult benchmark problems in flows of incompressible fluids. The imposed velocity field at the top moving wall of the cavity drives a large recirculation region inside the cavity. With increasing the moving wall velocity, and hence the Re number, additional smaller recirculation zones appear in the corners of the cavity.

Computations have been performed on a 60×60 macro element mesh with a ratio of 20 between the largest and smallest element lengths, and in the flow kinematics 4 subdomains have been used. The value of Reynolds number was selected as 10,000. Comparison of the results obtained with the benchmark results of Ghia [7] show very good agreement.

5.2 Flow in L shaped driven cavity

The flow in an L shaped driven cavity case is caused by the imposed velocity field at the top of the cavity, which is due to the action of viscous forces that drives the recirculating flow inside the lower right hand side of the cavity.

The computational mesh consisted of 300 equally distributed macro elements and 3 subdomains. Computations were performed for Re values of 400 and 1000. Here, the Reynolds number is defined as (\( h \) stands for the total height of the cavity wall),

\[
Ra = \frac{v_{\text{top}} h}{\nu}.
\]

(7)
The results for both the values of Re show that the flow pattern in the cavity consists of several vortices, the main one in the right upper corner. With growing Re numer value the left and the bottom vortex are becoming stronger and larger. Similar results were obtained by Perng [8].
5.3 Flow over a backward facing step

In the backward facing step flow case, the motion of the fluid in the channel is caused by the known inlet velocity profile at the left with all other boundaries as impermeable except for the outlet at the right. The computational mesh consists of 2900 macro elements with ratio 2 in the \( y \) direction and 15 subdomains.

Computations for a Re value of 800 was performed, where \( \text{Re} = \frac{\bar{v}H}{\nu} \) with \( \bar{v} \) being the average inlet velocity and \( H \) the full channel height.
Figure 8: Flow over a backward facing step: streamlines (top) and vorticity isolines (bottom) for Re=800.

In Table 1, comparison of the computed recirculation lengths is given for various approximation methods and the experimental results of Armaly [9], from where also the positions of $P_i$'s are taken.

Table 1: Backward facing step flow: comparison of recirculation lengths.

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>$P_1$</td>
<td>6.07</td>
<td>5.70</td>
<td>5.80</td>
<td>7.2</td>
</tr>
<tr>
<td>$P_2$</td>
<td>4.83</td>
<td>4.65</td>
<td>4.60</td>
<td>5.3</td>
</tr>
<tr>
<td>$P_3$</td>
<td>10.47</td>
<td>9.20</td>
<td>10.32</td>
<td>9.4</td>
</tr>
</tbody>
</table>

6 Conclusions

In this paper a novel computational algorithm based on the Boundary Domain Integral Method is presented. It consists of a segmentation technique, applied to a vector-potential formulation of the flow kinematics equation, and the macro-element approach with diffusion-convection fundamental solution for the computation of convection dominated flow kinetics. The resulting numerical algorithm proved to be stable and accurate in computing high Re flows, including driven cavity flow at Re=10,000 and the backward facing step flow at Re=800.

References


