Non Newtonian driven cavity flow comparison between boundary element and finite difference methods
Jure Marn, Leopold Škerget and Marjan Delić
University of Maribor, Faculty of Mechanical Engineering, Smetanova 17, 2000 Maribor, Slovenia

Abstract

An important class of fluids, namely generalized Newtonian fluids, have been the subject of considerable research in the area of petroleum engineering and biomechanics.

This contribution deals with a comparison between velocity-vorticity formulation used in the Boundary element method and the Finite difference method. Both methods decompose material properties into its constant contribution. The formulation is applied in its two dimensional form for flow in a driven cavity, believed to be a widely used benchmark problem for a wide array of fluid dynamics problem.

Introduction

A number of fluids are used within process engineering, among them fluids which do not obey Newton’s viscous law, the so called non Newtonian fluids. Among non-newtonian fluids oil, emulsion (mixture of oil with water), and human blood are most notable.

Regarding the numerical analysis of non Newtonian fluids, Bird and Wiest (1996) have presented a summary of known methods of calculations of various classes of non Newtonian
fluids, ranging from generalized Newtonian fluids to shear fluids with temporary changing material properties.

In addition, Vorwerk et al. (1994) obtained several interesting results comparing models of material properties between existing material property laws (notably power law, Carreau, and Ellis fluids) and their own experimental results.

In the area of computational techniques a number of quality presentations exist. For example, in the area of finite difference modelling, the work of Minkowycz et al. (1988) should be noted whereas in the area of boundary element methods Škerget et al. (1992) presents both theory and practical applications.

**Governing equations**

**General**

One starts from the equation of fluid motion for viscous, incompressible fluids

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

\[ \frac{Dv_i}{Dt} = - \frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{1}{\rho} \frac{\partial \tau_{ij}}{\partial x_j} \]  

(1)

whereas \( v_i \) stands for velocity, and \( x_i \) for position vectors, \( t \) for time, \( p \) for pressure and \( \rho \) for density, and \( \tau_{ij} \) for stress tensor as follows:

\[ \tau_{ij} = 2\nu \sigma_{ij} \]  

(2)

with \( \nu \) denoting kinematic viscosity. Note that \( s_{ij} \) stands for rate-of-strain tensor as follows:

\[ s_{ij} = \frac{1}{2} \left( \frac{\partial v_i}{\partial x_j} + \frac{\partial v_j}{\partial x_i} \right) \]  

(3)

All of the preceding equation are combined in the well known expression
suitable for further manipulation.

**Generalized Newtonian fluid models**

The equation (2) can be presented in various forms. For example, should Newton's viscous law apply, the equation (4) reduces to Navier-Stokes momentum conservation equation with material properties of the fluid summarized in dynamic viscosity, \( \mu = \nu \rho \). In order to model different fluids, one may relate rate-of-strain to variable viscosity, \( \eta \), using various functions. This approach is valid for common class steady state shear flows, known as generalized Newtonian fluids.

Hence, for time independent and for slowly varying shear flows one can use the generalized Newtonian fluid model as follows:

\[
\tau_{ij} = 2\eta(\dot{\gamma})s_{ij} 
\]

(5)

where \( \dot{\gamma} \) is function of \( s_{ij} \), or \( \dot{\gamma} = \sqrt{2s_{ij}s_{ij}} \).

Several empirical expressions for non-Newtonian viscosity (i.e. variable viscosity) have been suggested for the expression in eq. (5). However, for practical reasons, only three models will be used herewith, taken from Vorwerk et al. (1994) and Bird and Wiest (1996).

- **power law fluids**

This form is widely used for many industrial problems in which the plateau region of the \( \eta(\dot{\gamma}) \) curve is one of little importance as many applications involve high shear rates hence making the rapidly increasing part of the curve interesting. Most texts (see Bird and Wiest, 1996) list the curve as log \( \eta \) vs log \( \dot{\gamma} \) therefore prompting the empiricism in the form of

\[
\eta = m\dot{\gamma}^{n-1}
\]

(6)
with \( m \) and \( n \) determined for each fluid. Generally, \( n \) less than unity holds for pseudoplastic fluids whereas \( n \) equal to unity and \( m \) equal to \( \mu \) signifies Newtonian fluids.

Manipulation of eq. (6) shows numerical difficulties prompting different manifestation, namely

\[
\eta(\dot{\gamma}) = \begin{cases} 
\zeta_0 \left( \frac{\dot{\gamma}}{\dot{\gamma}_0} \right)^{n-1} & \dot{\gamma} \leq \dot{\gamma}_0 \\
\zeta_0 \frac{\dot{\gamma}}{\dot{\gamma}_0} & \dot{\gamma} > \dot{\gamma}_0
\end{cases}
\]  

(7)

where \( \zeta_0 \) stands for numerical constant. Note that if one deals with isothermal flows \( m \) and \( n \) must be known as functions of temperature.

- **Carreau fluids**

This model consists of four parameters, namely time constant \( \lambda \), the zero-shear-rate variable viscosity \( \eta_0 \), zero-shear-rate variable viscosity \( \eta_\infty \), and parameter \( n \) with the same meaning as in the power law fluid model:

\[
\eta(\dot{\gamma}) = \eta_\infty + (\eta_0 - \eta_\infty) \left[ 1 + (\lambda \dot{\gamma})^2 \right]^{(n-1)/2}
\]  

(8)

Many polymer solutions and melts were successfully described using this model.

- **Ellis fluids**

This model consists of three parameters, namely the zero-shear-rate variable viscosity \( \eta_0 \), parameter \( \alpha \), and parameter \( \tau_{1/2} \):

\[
\eta(\kappa) = \frac{\eta_0}{1 + (\tau/\tau_{1/2})^{\alpha-1}}
\]  

(9)
Here, \( \tau_{1/2} \) is the value of \( \tau \) at which \( \eta = \eta_0/2 \). Further, on a log-log plot in which material properties are usually given, \( \alpha \) is the slope of \( (\eta/\eta_0)-1 \) vs. \( \tau \).

**Velocity-vorticity formulation**

As the subject of this contribution are two dimensional flows, it is convenient to find a solution procedure (a) reducing conservation of momentum equation to its scalar form, and (b) eliminating pressure from the resulting equation. Applying \( \text{curl} \) to velocity, mass conservation, and momentum equations one finds

\[
\omega_i = e_{ijk} \frac{\partial v_k}{\partial x_j}
\]  

and

\[
\frac{\partial^2 v_i}{\partial x_j \partial x_j} + e_{ijk} \frac{\partial \omega_k}{\partial x_j} = 0
\]  

\[
\rho \frac{D\omega_i}{Dt} = \frac{\partial}{\partial x_j} \left[ \eta \left( \frac{\partial \omega_i}{\partial x_j} \right) \right] + \rho \frac{\partial \omega_j v_i}{\partial x_j} + \frac{\partial \tau_{ij}}{\partial x_j}
\]

where the last equation is known as a vorticity transport equation. As this contribution is concerned with two dimensional applications, the equations reduced to appropriate form will be discussed below.

**Boundary conditions and geometrical domain**

Driven flow in a cavity is a widely accepted benchmark problem consisting of a bulk flow flowing past cavity in which fluid rotates due to shear exerted by the bulk of flow. It is interesting to note that Bird and Wiest (1996) do not list this problem among several shown.

The boundary conditions are invariant with regard to the numerical method chosen. On the top of the cavity, the
velocity equals that of the bulk flow. On the walls, a no-slip condition is assumed. Hence, all boundaries are defined using Dirichlet conditions.

More generally, the treatment of the boundary conditions varies according to the method used. For example, in the case of boundary elements, two kinds of boundary conditions are specified, Dirichlet conditions (i.e. function values) on the boundary $\Gamma_1$, and Neumann conditions (i.e. normal flux values) on the boundary $\Gamma_2$.

As the velocity-vorticity formulation was chosen for actual solution of the problem as posed, vorticity needs to be specified on the boundary as well. The boundary values for vorticity are determined using the definition of vorticity, and are calculated from previously (i.e. previous computational step) known values of the velocity components in their two dimensional representation.

**Numerical solution**

Reducing the vorticity transport equation to its two dimensional form, one finds:

$$\rho \frac{D\omega}{Dt} = \frac{\partial}{\partial x_j} \left[ \eta(\dot{y}) \frac{\partial \omega}{\partial x_j} \right] + \frac{\partial f_j}{\partial x_j}$$

(13)

where $f_j$ stands for:

$$f_j = \begin{bmatrix} \frac{\partial \eta(\dot{y})}{\partial x} \left( \frac{\partial v_x}{\partial y} + \frac{\partial v_y}{\partial x} \right) - 2 \frac{\partial \eta(\dot{y})}{\partial y} \frac{\partial v_x}{\partial x} \\ \frac{\partial \eta(\dot{y})}{\partial y} \left( \frac{\partial v_x}{\partial y} + \frac{\partial v_y}{\partial x} \right) + 2 \frac{\partial \eta(\dot{y})}{\partial x} \frac{\partial v_y}{\partial y} \\ 0 \end{bmatrix}$$

(14)

in its Cartesian formulation.
**Boundary element method**

BEM version of this work is based on integral representation of the governing equation. The development of the algorithm is first achieved by decomposing the equations into kinematic and kinetic part, shown above in eqs. (11) and (12), respectively.

After applying the false transient scheme and transforming eqs. (11) and (12) into their respective suitable integral representation, one obtains

\[ c(\xi)v_i(\xi) + \int_{\Gamma} v_i \frac{\partial \mathbf{u}^*}{\partial n} d\Gamma = \int_{\Gamma} \frac{\partial v_i}{\partial n} \mathbf{u}^* d\Gamma + e_{ij} \int_{\Gamma} \omega n_j \mathbf{u}^* \]

for a kinematic part, and similarly,

\[ c(\xi)\omega(\xi) + \int_{\Gamma} \omega \frac{\partial \mathbf{u}^*}{\partial n} d\Gamma = \frac{\rho}{\mu} \int_{\Gamma} \left( \eta(\dot{\gamma}) \frac{\partial \omega}{\partial n} - \omega \nu_n + f \nu_j \right) \mathbf{u}^* d\Gamma + \]

\[ \frac{\rho}{\mu} \int_{\Omega} \left( \omega \nu_j - \eta(\dot{\gamma}) \frac{\partial \omega}{\partial x_j} - f \right) \frac{\partial \mathbf{u}^*}{\partial x_j} d\Omega + \beta \int_{\Omega} \omega \mathbf{u}^* d\Omega \]

for a kinetic part. Following that, the equations are written in a discretised manner (see Škerget and Hriberšek 1998) and solved.

**Finite difference method**

FDM version of this work first decomposes the flowfield and material properties into Newtonian and non Newtonian contributions:
\[ v_i = \bar{v}_i + v_{i,N} \]
\[ \omega = \bar{\omega} + \omega_N = e_{ijk} \frac{\partial \bar{v}_k}{\partial \bar{x}_j} + e_{ijk} \frac{\partial v_{k,N}}{\partial \bar{x}_j} \]  \hspace{1cm} (17)

\[ \eta(\dot{\gamma}) = \mu + \mu_N \]

where N denotes the non-Newtonian contribution. Applying the decomposed vorticity definition to the governing equations, one obtains the following system of equations:

\[ \rho \frac{D\bar{\omega}}{Dt} = \mu \frac{\partial^2 \bar{\omega}}{\partial \bar{x}_j \partial \bar{x}_j} \]  \hspace{1cm} (18)

\[ \rho \frac{D\omega_N}{Dt} = \frac{\partial}{\partial \bar{x}_j} \left[ \mu_N \left( \frac{\partial \bar{\omega}}{\partial \bar{x}_j} + \frac{\partial \omega_N}{\partial \bar{x}_j} \right) + \mu \frac{\partial \omega_N}{\partial \bar{x}_j} \right] + \frac{\partial f_{j,N}}{\partial \bar{x}_j} \]  \hspace{1cm} (19)

\[ f_{j,N} = \begin{bmatrix} 0 \\ -\frac{\partial \mu_N}{\partial \bar{y}} \left( \frac{\partial v_x}{\partial \bar{y}} + \frac{\partial v_y}{\partial \bar{x}} \right) + 2 \frac{\partial \mu_N}{\partial \bar{x}} \frac{\partial v_y}{\partial \bar{y}} \\ 2 \frac{\partial \mu_N}{\partial \bar{x}} \left( \frac{\partial v_x}{\partial \bar{x}} + \frac{\partial v_y}{\partial \bar{y}} \right) - 2 \frac{\partial \mu_N}{\partial \bar{y}} \frac{\partial v_x}{\partial \bar{y}} \end{bmatrix} \]  \hspace{1cm} (20)

These equations can be written in their dimensionless central difference form where

\[ \Delta x_+ = x^+(I+1,J) - x^+(I,J), \Delta x_- = x^+(I,J) - x^++(I-1,J), \]
\[ \alpha = \Delta x_+ / \Delta x_- \]
\[ \Delta y_+ = y^+(I,J+1) - y^+(I,J), \Delta y_- = y^+(I,J) - y^+(I,J-1), \]
\[ \beta = \Delta y_+ / \Delta y_- \]

and \( n \) denotes time step, \( Re \) stands for Reynolds number, I and J are indices in \( x \) and \( y \) direction, respectively. The
equations read (only eq. 18 shown for brevity):

$$\frac{\omega^{n+1}(I,J) - \omega^n(I,J)}{\Delta t} + \frac{1}{2} \left( v_x^{n+1}(I,J) \frac{\omega^{n+1}(I+1,J) + (\alpha^2 - 1)\omega^{n+1}(I,J) - \alpha^2 \omega^{n+1}(I-1,J)}{\alpha(\alpha + 1)\Delta x^*_x} \right) +$$

$$v_x^n(I,J) \frac{\omega^n(I+1,J) + (\alpha^2 - 1)\omega^n(I,J) - \alpha^2 \omega^n(I-1,J)}{\alpha(\alpha + 1)\Delta x^*_x} +$$

$$\frac{1}{2} \left( v_y^{n+1}(I,J) \frac{\omega^{n+1}(I,J+1) + (\beta^2 - 1)\omega^{n+1}(I,J) - \beta^2 \omega^{n+1}(I,J-1)}{\beta(\beta + 1)\Delta y^*_y} \right) +$$

$$v_y^n(I,J) \frac{\omega^n(I,J+1) + (\beta^2 - 1)\omega^n(I,J) - \beta^2 \omega^n(I,J-1)}{\beta(\beta + 1)\Delta y^*_y} =$$

$$\frac{1}{2 \text{Re}} \left( \frac{2}{\Delta x^*_x + \Delta x^*_y} \left( \frac{\omega^{n+1}(I+1,J) - \omega^{n+1}(I,J)}{\Delta x^*_x} - \frac{\omega^{n+1}(I,J) - \omega^{n+1}(I-1,J)}{\Delta x^*_x} \right) + \frac{\omega^n(I+1,J) - \omega^n(I,J)}{\Delta x^*_x} \right) \left( \frac{\omega^n(I,J) - \omega^n(I-1,J)}{\Delta x^*_x} \right) +$$

$$\frac{2}{\Delta y^*_x + \Delta y^*_y} \left( \frac{\omega^{n+1}(I,J+1) - \omega^{n+1}(I,J)}{\Delta y^*_y} - \frac{\omega^{n+1}(I,J) - \omega^{n+1}(I,J-1)}{\Delta y^*_y} \right) +$$

$$\frac{\omega^n(I,J+1) - \omega^n(I,J)}{\Delta y^*_y} \left( \frac{\omega^n(I,J) - \omega^n(I,J-1)}{\Delta y^*_y} \right) \right)$$

(21)

The solution is achieved using the following algorithm:

- solve the problem for the Newtonian fluid;
- start with iterations using false transient technique with appropriate relaxation parameter (e.g. Škerget and Hriberšek, 1998);
- within each iteration first calculate non Newtonian part of variable viscosity using velocity components from the last calculation;
- calculate the decomposed part of vorticity field;
- calculate the velocity field from the definition of vorticity;
- check for convergence rate, and if not satisfactory, complete another iteration.

**Concluding remarks**

This work is currently in progress and the latest results will be presented to elicit comments. Short of showing actual results, this contribution presents a comparison of two widely
used numerical models, i.e. boundary elements and finite differences to calculate two dimensional flow of non-Newtonian fluids in driven cavity.

The integral and differential representations of two dimensional velocity-vorticity formulation are presented, and three models for generalized Newtonian fluids shown along with outline of algorithm used for calculation of driven cavity flow.

**References**


