An incompressible magnetohydrodynamics solver

N. Aslan

Yeditepe University, Physics Department Kayisdagi, Istanbul, Turkey

Received: 3 August 2009/Accepted: 23 February 2010/ Published: 20 March 2010

Abstract

A two-dimensional nonlinear incompressible magneto-hydrodynamics (MHD) code is presented to solve steady state or transient charged or neutral convection problems. The flows considered are incompressible and the divergence conditions on the velocity and magnetic fields are handled by a relaxation scheme as pseudo-iterations between the real time levels. The numerical method takes the advantage of a matrix distribution scheme that runs on structured or unstructured triangular meshes. The time-dependent algorithm utilizes a semi-implicit dual time stepping technique with multistage Runge-Kutta algorithm. It is shown that the code developed here can be accurately used for the solutions of incompressible Navier Stokes or MHD equations.

PACs: 52.20. ± j; 41.20.Jb; 52.80. ± s; 03.50.De

Keywords: Plasma, Magnetohydrodynamics, Matrix distribution, Dual time stepping, Artificial compressibility

1. Introduction

When a conducting fluid or plasma moves in magnetic and electric fields, currents are produced causing change in fluid motion and these fields. Magnetohydrodynamics (MHD) deals with the motion of such electrically conducting media in the presence of electric and magnetic fields. In the incompressible-MHD approximation the plasma is assumed to be non-relativistic, low frequency moving as a single fluid with scalar pressure and temperature. The need for accurate numerical methods to model such flows with complex geometries motivated the recent developments in matrix distribution schemes [1]. When the flows are incompressible, one does not need upwind techniques with wave models to be used with fluctuation splitting. In the matrix distribution schemes, the equations are discretized such that the distribution matrix components are distributed to the nodes of the numerical grid made of unstructured triangles. The second order accuracy in space and time is achieved without the need of neighboring nodes. The paper presents a visual (running interactively with user) two-dimensional MHD code that is able to solve steady state or transient magnetized flows including heat transfer effects. Since the flow is incompressible and no magnetic monopoles exist, these conditions, i.e., \( \nabla \cdot \vec{V} = 0 \), \( \nabla \cdot \vec{B} = 0 \) are handled by relaxation schemes in the form of pseudo-iterations between the real time levels, see [2].

2. Incompressible magnetohydrodynamic equations

The Navier-Stokes (NS) equations for a conductive fluid are given by the following continuity, momentum, and energy equations:

\[
\nabla \cdot \vec{V} = 0
\]

\[
\rho \left[ \frac{\partial \vec{V}}{\partial t} + \vec{V} \cdot \nabla \vec{V} \right] + \nabla P = \mu \nabla^2 \vec{V} + \rho \vec{g} + \vec{F}_{EM}
\]

(2)

\[
\rho C_v \left( \frac{\partial T}{\partial t} + \vec{V} \cdot \nabla T \right) = k \nabla^2 T + \frac{\vec{F}_{EM} \cdot \vec{r}}{\sigma}
\]

(3)

where \( \rho \) is the density, \( \vec{V} \) is the velocity, \( P \) is the scalar pressure, \( T \) is the temperature, \( \vec{F} \) is the total current density. In addition, \( \mu \) and \( \vec{g} = -g \vec{e}_y \) are viscosity and gravitational acceleration, \( C_v \) is the specific heat, \( k \) and \( \sigma \) are thermal and electrical conductivities, respectively. \( \vec{F}_{EM} \) is the electromagnetic Lorentz force given by:

\[
\vec{F}_{EM} = \rho_e \vec{E} \times \vec{B} + \vec{J} \times \vec{B}
\]

(4)

where \( \rho_e \) is the charge density, \( \vec{E}, \vec{B} \) are total electric and magnetic fields and \( \vec{J} \) is the total current density. Note that here “total” means the sum of internal and external. Note also that the last term in Eq. (3) denotes the Joule heating due to the currents. Since the fluid is assumed to be conductive, the NS equations must be
In this work, these conditions are modified to read accompanied with the Maxwell’s equations given as:

\[
\nabla \times \vec{E} = -\frac{\partial \vec{B}}{\partial t}, \tag{5a}
\]

\[
\nabla \times \vec{B} = \mu_0 \vec{J} + \mu_0 \varepsilon_0 \frac{\partial \vec{E}}{\partial t} \approx \mu_0 \vec{J}, \tag{5b}
\]

\[
\n\nabla \cdot \vec{E} = \frac{\rho_s}{\varepsilon_0} = 0, \tag{5c}
\]

\[
\n\nabla \cdot \vec{B} = 0, \tag{5d}
\]

where \(\varepsilon_0\) and \(\mu_0\) are the dielectric constant and magnetic permeability, respectively and \(\vec{E}\) is the internal electric field. Since there is no charge separation in the fluid, \(\rho_s\) vanishes in the fluid.

The MHD equations (Eqs.1-5) derived here are used for incompressible flows, so that the divergence constraints for velocity and magnetic fields (i.e., \(\nabla \cdot \vec{V} = 0, \nabla \cdot \vec{B} = 0\)) should be satisfied. In this work, these conditions are modified to read as:

\[
\frac{\partial \rho}{\partial t} + \beta^2 \nabla \cdot \vec{V} = 0, \tag{6}
\]

\[
\frac{\partial \psi}{\partial t} + \alpha^2 \nabla \cdot \vec{B} = 0, \tag{7}
\]

where \(\tau\) is the pseudo-time step, \(\beta^2\) is the artificial compressibility parameter, \(\alpha^2\) is the artificial magnetic monopole parameter and \(\psi\) is the magnetic monopole function which is used to correct the magnetic fields. These equations are solved as sub-iterations between each time step to force the velocity and magnetic fields to satisfy the divergence constraints. As sub-iterations converge (i.e., \(\partial / \partial \tau \to 0\)) \(P\) and \(\psi\) relax to the necessary distributions which are able to correct the velocity and magnetic fields in such a way that the divergence conditions are satisfied. In order this procedure to be consistent, with the relaxation scheme described here, it can be shown that the terms: \([1/\rho\mu_0 \cdot \nabla \vec{B}, \nabla \psi]\) and \([\nabla \psi]\) should be added to the energy equation, Eq. (3) and to Faraday’s law Eq.(5a). By considering all the derivations so far, the resulting MHD equations excluding external fields but including the effects of \(\psi\) turn into the following set of equations in cartesian geometry:

\[
\frac{\partial \rho}{\partial t} + \beta^2 \nabla \cdot \vec{V} = 0, \tag{8}
\]

\[
\frac{\partial \psi}{\partial t} + \alpha^2 \nabla \cdot \vec{B} = 0, \tag{9}
\]

\[
\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + \frac{1}{\rho} \frac{\partial p}{\partial x} + \frac{\partial}{\partial x} \left( \frac{\partial \psi}{\partial x} - \frac{\partial \psi}{\partial y} \right) = \nu \nabla^2 u, \tag{10}
\]

\[
\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + \frac{1}{\rho} \frac{\partial p}{\partial y} - \frac{B_x}{\mu_0} \left( \frac{\partial \psi}{\partial x} - \frac{\partial \psi}{\partial y} \right) = \nu \nabla^2 v - g, \tag{11}
\]

\[
\frac{\partial B_x}{\partial t} + u \frac{\partial B_x}{\partial x} + B_x \frac{\partial u}{\partial x} - B_y \frac{\partial u}{\partial y} + v \frac{\partial B_y}{\partial x} + \frac{\partial B_y}{\partial y} - \frac{d \psi}{d x} = \frac{1}{\mu_0} \nabla^2 B_x, \tag{12}
\]

\[
\frac{\partial B_y}{\partial t} + B_y \frac{\partial u}{\partial y} + B_x \frac{\partial v}{\partial x} + \frac{d \psi}{d x} + \frac{d \psi}{d y} = \frac{1}{\mu_0} \nabla^2 B_y, \tag{13}
\]

\[
\frac{\partial \tau}{\partial t} + u \frac{\partial \tau}{\partial x} + v \frac{\partial \tau}{\partial y} + \frac{B_x}{\rho c_v} \frac{\partial B_y}{\partial x} + \frac{B_y}{\rho c_v} \frac{\partial B_y}{\partial y} = \frac{k}{\rho c_v} \nabla^2 \tau + \frac{\sigma (u B_y - v B_x)^2}{\rho c_v}. \tag{14}
\]

Note that the continuity equation, Eq. (1), states that the density is constant. However, when temperature gradients exist in the solution domain, the natural convection occurs with slight changes in the density. In this work, it is assumed that the density variations are very small so that the Boussinesq approximation holds. In this case, the density can be written as a function of temperature as follows:

\[
\rho = \rho_\infty [1 - \beta (T - T_\infty)]. \tag{15}
\]

where \(\rho_\infty\) and \(T_\infty\) are ambient density and temperature (usually taken as outside parameters) and \(\beta\) is the expansion coefficient. Inserting this form of density to the right hand side of the momentum equations, and defining a new pressure as \(P = P' + P_{\infty}\) (where \(P'\) is the reduced pressure) the \(y\) component of the momentum equation becomes:

\[
\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + \frac{1}{\rho} \frac{\partial P'}{\partial y} - \frac{B_x}{\mu_0} \left( \frac{\partial \psi}{\partial x} - \frac{\partial \psi}{\partial y} \right) = \nu \nabla^2 u + g \beta (T - T_\infty). \tag{16}
\]
For the energy equation a new variable,  
\[ \theta = \frac{T - T_{\text{ref}}}{\Delta T} \]
(where \( \Delta T \) is the temperature difference) is defined and the heat equation (24) is rewritten as follows:

\[ \frac{\partial \theta}{\partial t} + u \frac{\partial \theta}{\partial x} + v \frac{\partial \theta}{\partial y} + \frac{\partial \psi}{\partial y} \left[ \frac{B_y}{\sigma} \right] + \frac{\partial \psi}{\partial x} \left[ \frac{B_x}{\sigma} \right] = \frac{k}{\rho c_p} \nabla^2 \theta + \frac{\sigma_m}{\rho c_p} \mu \nabla^2 \theta. \]  
(17)

In this work, the following dimensionless parameters were used in order to write the dimensionless form of MHD equations:

\[ \bar{x}' = \frac{x}{L_0}, \bar{t}' = \frac{t}{t_0}, \bar{P}' = \frac{P}{P_0}, \bar{\rho}' = \frac{\rho}{\rho_0}, \]
\[ \bar{B}' = \frac{B}{B_0}, \bar{\psi}' = \frac{\psi}{\psi_0}, \bar{E}' = \frac{E}{E_0}, \bar{\nabla}' = \frac{\nabla}{\nabla_0} \]

where the procedure of making the MHD equations dimensionless produces the following parameters:

Reynold’s Number \( Re = L_0 V_0 / \nu, \)  
(18)

Interaction Parameter \( N = \frac{\sigma B^2 L_0}{\mu_0 V_0^2} \)  
(19)

Magnetic Reynold’s Number \( Re_m = \mu_0 \sigma V_0 L_0 = \frac{\mu_0 V_0 L_0}{\eta}, \)  
(20)

Hartmann Number \( Ha = \sqrt{NR_e} = B_0 L_0 / \sqrt{\eta / \mu}, \)  
(21)

Rayleigh Number \( Ra = \frac{g \beta \Delta T L_0^3}{\nu c_p k}, \)  
(22)

Prandtl Number \( Pr = \frac{v}{k}, \frac{k}{\rho c_p}, \)  
(23)

Eckert Number \( E = \frac{v^2}{c_p k}, \)  
(24)

Using these parameters, the dimensionless form of the MHD equations can be cast into:

\[ \frac{\partial \bar{u}}{\partial \bar{t}} + I_m \frac{\partial \bar{u}}{\partial \bar{t}} + A \frac{\partial \bar{u}}{\partial \bar{x}} + B \frac{\partial \bar{u}}{\partial \bar{y}} = \bar{S}_e, \]  
(25)

where \( \bar{U} = [P', u, v, \theta, B_x, B_y, \psi] \) is the state vector, \( I_m = \text{diag} [0,1,1,1,1,1,0] \) is the diagonal matrix, \( A \) and \( B \) are coefficient matrices of \( \partial \bar{U} / \partial \bar{x} \) and \( \partial \bar{U} / \partial \bar{y} \), and \( \bar{S}_e \) and \( \bar{S'_e} \) are viscous and external sources. This equation in detail is given below:

\[ \begin{bmatrix} u \\ v \\ \theta \end{bmatrix} + \begin{bmatrix} P' \\ B_x \\ B_y \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \\ 0 \end{bmatrix} \]
\[ \frac{\partial}{\partial \bar{t}} + I_m \frac{\partial}{\partial \bar{t}} \left[ \begin{bmatrix} u \\ v \end{bmatrix} + \begin{bmatrix} P' \\ B_x \\ B_y \end{bmatrix} \right] + \begin{bmatrix} 0 \\ 0 \\ 0 \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \\ 0 \end{bmatrix} \]

where

\[ \bar{S}_e = \begin{bmatrix} 0 \\ 0 \\ 0 \end{bmatrix}, \bar{S'_e} = \begin{bmatrix} 0 \\ 0 \\ 0 \end{bmatrix} \]

Noting that the first and last elements of Eq. (26) are exactly the same as Eqs. (7) and (8) and that the time rates of \( P' \) and \( \psi \) are separated from original equations by means of \( I_m \).

3. Matrix distribution scheme

The incompressible MHD equations can be discretized by several discretization techniques (such as finite difference, finite volume, finite element etc.). In this work, “matrix distribution scheme” is used [1]. In this scheme, the discretized form of the system of MHD equations is obtained by integrating it over the area of a triangle, \( \Omega_T \), which has the nodes of i, j, k, see Fig. 1a:

\[ \iint_{\Omega_T} \left[ \frac{\partial \bar{U}}{\partial \bar{t}} + A \frac{\partial \bar{U}}{\partial \bar{x}} + B \frac{\partial \bar{U}}{\partial \bar{y}} \right] d\Omega_T = \iint_{\Omega_T} (\bar{S}_e + \bar{S'_e}) d\Omega_T. \]  
(27)

By using forward difference formula for the time derivative and employing Gauss law for the
convection terms on the left hand side of Eq. (27), one obtains the following update formula at node “i” which is located at one of the vertices of triangle “T”:

\[
\frac{\mathbf{U}_{i}^{n+1} - \mathbf{U}_{i}^{n}}{\Delta t} + \oint_{\partial T} \left( \mathbf{A}_{x} + \mathbf{A}_{y} \right) \mathbf{U} d\ell_{T} = \left( \mathbf{S}_{i}^{p} + \mathbf{S}_{i}^{h} \right) \Omega_{i},
\]

where \( \Omega_{i} \) is the Veronoi area surrounding node i (see Fig. 1) and \( \Delta t \) is the time interval. By using numerical trapezoidal method for the flux integral (contour integral in Eq. 28) and considering all the triangles surrounding node i, Eq. (28) can be written as:

\[
\frac{\mathbf{U}_{i}^{n+1} - \mathbf{U}_{i}^{n}}{\Delta t} = \frac{\Delta t}{\Omega_{i}} \left[ \sum_{\text{triangles around } i} \left( \Phi_{i}^{T} - \Omega_{i} \left( \mathbf{S}_{i}^{p} + \mathbf{S}_{i}^{ext} \right) \right) \right],
\]

for the state vector at the node “i”. As seen, the source terms are distributed equally among the nodes of “T” while \( \Phi_{i}^{T} \) is the cell fluctuation (a fraction of total fluctuation \( \Phi_{T} \)) assigned to the node “i” given by:

\[
\Phi_{i}^{T} = B_{i} \Phi_{T} = B_{i} \sum_{m=1}^{3} K_{m} \mathbf{U}_{m},
\]

where \( B_{i} \) is the distribution matrix which is responsible in determining how much of \( \Phi_{T} \) should be distributed among the nodes of “T”. Here, \( K_{m} = \left( A_{nx} + B_{ny} \right) n_{m}^{2}/2 \) is called the 2D Jacobian matrix (at node \( m = i, j, k \) of triangle T) and it is given by:

\[
K_{n} = \begin{bmatrix}
0 & \beta^{2} n_{x} & \beta^{2} n_{y} & 0 & 0 & 0 \\
\beta^{2} n_{x} & V_{x} & 0 & 0 & 0 & 0 \\
\beta^{2} n_{y} & 0 & V_{y} & 0 & 0 & 0 \\
-\beta^{2} n_{x} & -\beta^{2} n_{y} & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & \frac{EB}{V_{n}} \\
0 & 0 & 0 & 0 & 0 & 0 \\
\end{bmatrix},
\]

where \( n_{x} \) and \( n_{y} \) are the x and y components of the normal vectors (see Fig. 1) whose magnitudes are equal to the associated side lengths, \( V_{T} = \left| A_{nx} + B_{ny} \right| \) is the tangential velocity along streamlines. It should be noted here that the numerical integrations were performed by assuming that the state variables are assumed to vary linearly over the triangle.

The matrix \( B_{i} \) assumes the role of distributing appropriate fractions of the total fluctuation to the nodes of triangle. The different forms for \( B_{i} \) results in different methods all of which produce similar results. For example, when \( B_{i} = I/3 \), where I is the unit matrix, the method is called classical Galerkin Finite Element method which has some stability problems. In the method presented here, the classical second order accurate Lax-Wendroff method is used such that the distribution matrix (Galerkin form plus extra dissipation) is defined:

\[
B_{i} = \frac{1}{3} I + \frac{\Delta t}{2\Delta t} K_{i}^{T},
\]

where “\( \Delta t \)” is a characteristic local time step for the triangle considered and the second term is extra numerical dissipation. It is noted that, the careful design of this matrix is very important for numerical accuracy and that regardless of the method used, the distribution matrices on the nodes of “T” should satisfy the following property for consistency:

\[
B_{1} + B_{2} + B_{3} = I.
\]

When the updating procedure is performed triangle by triangle, the updating formula given by Eq. (29) turns into the following update at node i due to the triangle T:

\[
U_{i}^{n+1} = U_{i}^{n} - \frac{\Delta t}{\Omega_{i}} \left[ \Phi_{i}^{T} - \Omega_{T} \left( \mathbf{S}_{i}^{p} + \mathbf{S}_{i}^{ext} \right) \right] + tfot,
\]

where “tfot” represents the terms from other triangles surrounding node “i”. When all triangles are visited and their associated fluctuations are distributed to their nodes by means of the distribution matrix, all the nodes in the mesh will be updated at new time step.

4. Numerical results

Test 1. Channel flow along rectangular channel and exit through a small outlet

This non-magnetic test is a purely hydrodynamics flow through a channel which has a small outlet on the right. For this problem, \( Re = 10 \) was taken and a \( 61 \times 31 \) isotropic mesh (with 1891 nodes, 3600 triangles) was used for a domain of \( x, y \in \{0-2, 0-1\} \). The artificial compressibility parameter of 10 was used. The left boundary condition was taken as \( P = 0, V_{x} = 1, V_{y} = 0 \) while free outlet conditions for \( V_{x} \) and \( P \) was used and \( V_{y} = 0 \) was
taken for the right boundary. As seen from Fig. 2, the constant velocity profile at the inlet turns into a parabolic profile with an increasing velocity tip at the exit. Fig. 2b shows the residual of the solutions as a function of iteration number. As seen, the solution has a full convergence at iteration number of 700.

Fig. 2. Hydrodynamics flow in a rectangular channel which has a small outlet on the right

Test 2. Channel flow around obstacles within L shaped channel

In this test a L shaped channel (x=[0;10], y=[0;5]) which includes a semi-circle and circle obstacles (of radii 0.5) were considered in order to examine forced flow by pressure differences (higher on the left inlet boundary and lower on the right top boundary). When Re = 1000, the flow entering from the left boundary goes around the cylindrical obstacle creating circulating regions past the object and the left boundary near outlet. However, as Re = 10 is taken, a viscous and laminar flow develops around the semi circle obstacle creating no vortices along the channel. These results are represented in Fig. 2. In addition, for Re = 1000 case there also exists a circulation zone on one side of fluid near exit.

Test 3. Vertical flow through a narrow channel with a side exit

In this test, the fluid flows downward by the gravitational force and exits from a narrow outlet at the bottom left side of the channel. Free inlet and outlet boundary conditions were used at the top entrance and left exit respectively (P_{top}=0, P_{bottom} = -1) and side walls were considered to be no slip walls (i.e., velocity is set to zero). Fig. 4 shows the (21 x 41) structured mesh, velocity vectors and pressure field of this flow. As seen in this purely hydrodynamics flow (where electric and magnetic fields are negligible) the immediate parabolic profile of vertical flow turns right and exits leaving a circulation zone around the lower left corner.

Test 4. Incompressible low Re number flows around NACA-1240 airfoil

In this purely hydrodynamics test, the fluid flows right passing around a NACA-1240 airfoil. The resulting pressure distributions for Re = 20 and Re = 100 cases are shown in Fig. 5. As seen from these pressure distributions, the maximum pressure is exerted at the tips for both cases and the pressure on the lower boundary of airfoil is higher than that of upper boundary for Re = 100 case and almost the same for Re = 20 case.

Test 5. Thermal room flow with open upper boundary

In the following test the top boundary was taken as free boundary and different wall temperatures were considered. As the side walls are colder than the bottom temperature a simple circulation occurs while two circulation regions are observed as side walls are hotter than the bottom wall.
The flow enters from one side for the former case but it enters from the center for the latter. For this flow Re = 100 and 1000 was considered on a 41 × 41 isotropic mesh. The results are shown in Fig. 6.

Test 6. Thermal cavity flow

In this hydrodynamics test with heat transfer, the temperature equation was solved along with NS equation and Boussinesq approximation was used in order to couple temperature gradients into the density differences. The cavity flow originates due to the suddenly created hot right and cold left boundary at constant temperatures. The temperature gradient in x-direction combined with gravity in y-direction causes a circulation in the cavity. The normal derivative of temperatures were taken as zero at the top and bottom boundaries (isolated boundaries) while side walls were considered no slip walls at constant temperatures. As seen from Fig. 7, the thermal boundary layer gets narrower as Ra number is increased from (a) 1000 to (b) 10000.

Fig. 4. Vertical flow through a narrow channel with a side exit. (a) utilized isotropic mesh, (b) velocity profile, (c) pressure distribution.

Fig. 5. Pressure distributions of a incompressible flow around a NACA airfoil.

Fig. 6. Thermal room flow with open upper boundary, (a) heated from below, (b) heated from sides.
Test 7. Rayleigh Taylor instability

This test is about a heavy fluid supported against gravity by a lighter fluid which constitutes an instability: Rayleigh-Taylor instability (RT). As known, a liquid that is held inverted will cause the liquid to fall out the sides of the container, while a column of air rises along the center in its place. While the atmospheric pressure associated with the air column is sufficient to support the weight of the liquid, the instability is caused by tiny perturbations at the interface. More generally, the acceleration may be different from the earth’s gravity, or even impulsive where the resulting flow is called a Richtmyer-Meshkov instability. The interfacial perturbations may be of a single wavelength ($\lambda$), or comprise of a superposition of many waves (i.e. a spectrum). A single wavelength will grow exponentially in time, before saturating to a constant terminal velocity at late-time. Structures of the light fluid penetrating the heavy are called bubbles, while the corresponding fingers of the heavy fluid are termed spikes. When a spectrum of modes is present, the interactions between modes result in a turbulent flow, characterized by a high level of mixing between the fluids. The linear theory suggests that if a horizontal magnetic field ($B_x$) is applied, this instability can be suppressed. For this test, a Cartesian isotropic grid: $x = [0; 4]$, $y = [-3; 1]$ was taken and the horizontal boundary was located at $y = 0$. Initially, top and bottom densities were taken as 20 and 1, respectively and $P = 1$ was considered. Note
An incompressible …

that, the following equation was solved along with Navier Stokes equation to specify density differences:

\[ \frac{\partial \rho}{\partial t} + \mathbf{V} \cdot \nabla \rho = 0. \]

If no magnetic field is applied the perturbations grow exponentially producing downward spikes and upward bubbles but when a magnetic field of \( B_x = 2 \) is applied, the perturbations are suppressed and the interface structure is not destroyed as shown in Fig. 8.

Fig. 9. Conducting liquid metal flow past a circular cylinder within parallel magnetic field
Test 8. Liquid metal flow past circular cylinder

This problem describes conducting incompressible liquid metal flow past a circular cylinder under the effect of perpendicular magnetic field. The flow around cylinder becomes unstable and vortices start to shed from cylinder surface behind which periodic flow is obtained for both Re = 100 and Re = 400. It is shown in this test that such flows can be stabilized by using an external magnetic field. The cylinder with a radius of unity was placed at (x = 0, y = 0) in a computational domain of (-4.5 ≤ x ≤ 15.5, -4.5 ≤ y ≤ 4.5). The inflow boundary condition was used on the inlet (left surface) with u = 1, v = 0, \( B_y = 1 \) and reflecting boundary conditions were used on upper and lower boundaries with v = 0, \( B_y = 1 \). The right boundary was chosen as outgoing boundary on which no condition imposed on velocity but \( B_y = 1 \) was taken when magnetic fields exist. The cylinder surface is taken as no slip boundary with u = v = 0 and \( B_y = 1 \). This problem was run for no magnetic field, with a perpendicular field (in y direction) for Re = 100 and Re = 400. The interaction parameters were chosen as N = 0, 0.001, 0.0625, 0.25 corresponding to Hartmann numbers of Ha = 0, 1, 2.5, 5.0, respectively for magnetic Reynolds number of 10^{-5}. Fig. 9 shows the time behaviour of x-velocity contours at times t = 30, 90, 120 for Re = 100 and Re = 400 with no magnetic field. When \( B_x = 0 \) and \( B_y = 1 \) is used as a boundary conditions, this field eliminates the transient oscillations and finally periodic behaviour when Hartmann number is high enough and Re number is not greater than 100. As seen the vertical magnetic field totally eliminates oscillations for Re = 100 but does not work efficiently for Re = 400. These results show that the addition of magnetic field in a conducting medium adds viscosity and creates an extra drag on the immersed body. These effects are of great importance for metal casting in atmospheric pressures and thin film deposition in vacuum chambers.

5. Conclusion

A matrix distribution scheme with dual time stepping operating on structured or unstructured triangular mesh was presented to solve Navier Stokes and Magnetohydrodynamic equations to examine heat transfer and magnetization effects. Boussinesq approximation was used in order to represent the buoyancy effects of temperature. The code developed here is visual and user friendly letting user to follow flow features during solutions. It is shown by numerical results that the code is reliable and robust and it can be used safely for a variety of two dimensional problems varying from simple hydrodynamics to magnetized flows under the effects of external and internal fields and sources. Currently, three dimensional form of this code is being developed by the author.

References