Study of incompressible MHD flow in a circular pipe with transverse magnetic field using a spectral/finite element solver.

X. Dechamps * and G. Degrez †

Université Libre de Bruxelles, Av. F-D. Roosevelt 50 (CP 165/41), Brussels, Belgium

M. Rasquin ‡

Argonne Leadership Computing Facility, Argonne National Laboratory, Argonne (IL), USA

K.E. Jansen §

Department of Aerospace Engineering, University of Colorado Boulder, Boulder (CO), USA

This work is concerned with the numerical study of an unsteady and incompressible flow of an electrically conducting liquid metal inside a pipe of circular cross-section at low Reynolds number. In particular, the aim of this work is to understand better the influence of an external transverse magnetic field on the flow field and the transition between a fully turbulent regime to a fully laminar regime at moderate Reynolds number as the Hartmann increases. For that purpose, direct numerical simulations have been carried out and provide mean velocity profiles, turbulence intensity profiles, physical and numerical dissipation profiles and skin friction data. The obtained results show the damping of turbulence due to the action of the magnetic field and the development of anisotropy in the flow.

Nomenclature

\( \mu_0 \) magnetic permeability of vacuum
\( \nu \) kinematic viscosity
\( \phi \) electric potential
\( \rho \) density of the fluid
\( \sigma \) electrical conductivity of the fluid
\( B \) external magnetic field
\( J \) current density
\( u \) velocity field = \((u_z, u_r, u_\theta)\)
\( Ha \) Hartmann number
\( p \) kinematic pressure \(P/\rho\)
\( Re \) hydrodynamic Reynolds number
\( Re_m \) magnetic Reynolds number
\( u_b \) mean streamwise velocity

---

*Graduate student, department of Aero-Thermo-Mechanics, Université Libre de Bruxelles, Av. F-D. Roosevelt 50 (CP 165/41), Brussels, Belgium, AIAA member.
†Professor, department of Aero-Thermo-Mechanics, Université Libre de Bruxelles, Av. F-D. Roosevelt 50 (CP 165/41), Brussels, Belgium, AIAA member.
‡Research associate, Argonne Leadership Computing Facility, Argonne National Laboratory, Argonne (IL), USA.
§Professor, department of Aerospace Engineering, University of Colorado Boulder, Boulder (CO), USA.
I. Introduction

The study of duct flow for electrically conducting liquid metal fluids subject to an externally applied magnetic field is acknowledged to be a good approach for a better understanding of the fundamental properties of magnetohydrodynamic (MHD) turbulence. This fundamental type of flow is encountered frequently in industrial processes such as the casting/stirring of steel or the Czochralski process used to obtain single crystals of semiconductors.

The experimental study of the MHD pipe flow began in 1937 with Hartmann. In the 60’s-70’s, numerous experiments were conducted on several configurations (non conductive wall or with finite conductivity - transverse or longitudinal magnetic field). In the late 90’s and early 2000’s, numerical results began to appear. Finally, in 2008, additional experimental results were obtained for a Reynolds number $Re_b = 11,300$. Overall, little information is available concerning the properties of turbulence in the case of a non-conducting pipe flow under a transverse magnetic field. The available results are generally limited to the velocity profiles, velocity fluctuations and global skin friction coefficient. Up to now, no relevant data about the physical characteristics of turbulent structures are available and the few available numerical results are restricted to low Reynolds numbers.

In contrast, many more results were obtained in the case of a flow inside a channel or a duct with rectangular/square cross-section. This is linked with the greater choice of the electrical conductivity of the walls and the different resulting physical phenomena. Indeed, through a given combination of electrically conducting/non-conducting walls, jets can for instance appear along sidewalls. These jets are thus source of flow instabilities that do not occur (or not so easily) in the case of the pipe flow. Due to this increasing interest of the scientific community to duct flows, numerical simulations at higher Reynolds numbers were performed. Ref. 24 for instance considered a Reynolds number equal to $10^5$ and showed a quasi-laminar core in the center of the square duct and in the Hartmann layers, with still a turbulent flow near the walls parallel to the magnetic field (Shercliff layers), a flow pattern that still remains to be observed in the case of pipe flows.

One main objective of this study is to improve our understanding of turbulent MHD pipe flows subject to a uniform transverse magnetic field and the transition between a fully turbulent regime to a fully laminar regime at moderate Reynolds numbers as the Hartman number increases. The bulk Reynolds number $Re$ is chosen equal to 8000 and the Hartmann number $Ha$ ranges between 0 and 100. This range allows to cover the turbulent and laminar regimes with the laminarization process being linked with the Joule dissipation.

The physical model is discussed in section II, whereas the numerical method is presented in section III. The numerical results obtained from direct numerical simulations are discussed in section IV and include the velocity profiles, velocity fluctuations, skin friction coefficient and physical/numerical dissipations. Finally, the conclusions are presented in section V.

II. Physical model

This work is concerned with the unsteady, incompressible and isothermal flow of an electrically conducting and Newtonian fluid (i.e. a liquid metal) inside a pipe of circular cross-section and diameter $d = 2R$ (see figure 1). An external, constant and uniform magnetic field $B = B\hat{e}_x$ is applied along the transverse direction $x$ ($\theta = 0$). The region near the wall in the direction parallel to $B$ is referred to as the Hartmann layer and the one in the direction perpendicular to the magnetic field is referred to as the Roberts layer. The wall of the cylinder is solid, smooth and electrically insulated. Periodic conditions are imposed at the inlet ($z = 0$) and the outlet ($z = L_z$) of the domain. Under the flow conditions specified later, we observe that the magnetic Reynolds number $Re_m = \mu_0 \sigma_0 u_b d \ll 1$. Under this condition, the inductionless form of the MHD equations described by the set of equations Eqs. (1) can be used. As a consequence of the assumption $Re_m \ll 1$, the contribution of the induced magnetic field is negligible in comparison with the external magnetic field $B$. Another way to interpret this assumption is that the velocity field is influenced by the magnetic field but the opposite is not true. Eqs. (1) are respectively the continuity equa-
tion, the momentum equations, the conservation of charge $\nabla \cdot J = 0$ and the definition of the current density through Ohm's law. The validity of this model has been confirmed in numerous works (see e.g. Refs. 2,25). Additionally, numerical results of Knaepen\textsuperscript{26} show that no significant deviations are observed for $Re_m \leq 1$ between the inductionless model and the full set of MHD equations. The unknowns of the problem are the components of the velocity field $u$, kinematic pressure $p = P/\rho$ and the electric potential $\phi$. In the momentum equations, the Lorentz force $f_L = J \times B$ creates a link between the flow and the magnetic field. This force is generally opposed to the flow and tends to flatten the velocity profile.

\begin{align*}
R_{cont} &= \nabla \cdot u = 0 \quad (1a) \\
R_{mom} &= \frac{\partial u}{\partial t} + (u \cdot \nabla)u + \nabla p - \nu \nabla^2 u - \frac{J \times B}{\rho} = 0 \quad (1b) \\
R_{charge} &= \nabla^2 \phi - \nabla \cdot (u \times B) = 0 \quad (1c) \\
J &= \sigma (-\nabla \phi + u \times B) \quad (1d)
\end{align*}

The set of boundary conditions specified in Eqs.(2) are applied along the wall in order to close the problem ($n$ stands for the normal to the wall). Regularity conditions are imposed on the axis ($r = 0$) for velocity, pressure and electric potential (see section III.A).

\begin{align*}
\bar{u}(z, r = R, \theta) &= 0 \quad (2a) \\
\frac{\partial \phi}{\partial n}(z, r = R, \theta) &= 0 \quad (2b)
\end{align*}

The non-dimensional parameters that characterize the problem are

\begin{align*}
Re_b &= \frac{\text{Inertial forces}}{\text{Viscous forces}} = \frac{u_b d}{\nu} \quad \text{Reynolds number} \quad (3a) \\
Ha &= \frac{\text{Lorentz force}}{\text{Viscous forces}} = BR \sqrt{\frac{\sigma}{\rho \nu}} \quad \text{Hartmann number} \quad (3b)
\end{align*}

### III. Numerical method

The unsteady equations (1) are solved by the in-house hybrid Spectral/Finite Element LES algorithm (SFELES). The spatial discretization in SFELES is based on a stabilized finite element method in 2D planes, whereas a collocated spectral method describes the problem in the out-of-plane direction. The stabilization of the 2D finite element method is ensured by the traditional Streamline Upwind Petrov-Galerkin (SUPG) and Pressure-Stabilized Petrov-Galerkin (PSPG) formulation\textsuperscript{27,28}. The main assumption in SFELES relies therefore on a direction of periodicity in the flow and can simulate 3D flows associated with complex planar of axisymmetric geometries. Several benefits result from this combination of discretizations:

- The in-plane finite element method is well suited for unstructured meshes of complex 2D geometries;
- Memory requirements are reduced because only a single 2D mesh needs to be stored (connectivity, coordinates, etc);
- As it will be demonstrated below, a pseudo-spectral treatment of the non-linear terms leads to a decoupling of the discretized equations for each Fourier mode. For each time step, a set of independent 2D linear systems are solved in Fourier space, which is a natural way to introduce a first level of parallelism.

This pseudo-spectral approach implies the evaluation of the non-linear terms in the physical domain and their transfer back to Fourier space with a Direct Fourier Transform. In order to avoid aliasing problems during the Direct Fourier Transforms, only $N_{comp} = 2/3N$ modes are kept and effectively computed in Fourier space. Finally, the full 3D solution in physical space is reconstructed from an Inverse Fourier Transform applied to the Fourier modes computed in Fourier space. Although 1/3 of the modes are thrown out during the Direct Fourier Transform, the solution in physical space is evaluated at $N$ locations in the periodic direction (i.e. $N$ 2D planes).
III.A. Spatial discretization

Considering the axisymmetric version of SFELES, the unknowns \( q = \{ u_z, u_r, u_\theta, p, \phi \} \) in the planes \( \theta = cst \) are represented by linear, triangular elements \( q(z, r, \theta, t) = \sum_{j=1}^{N_{nodes}} q_j(\theta, t) N_j(z, r) \) where the index \( j \) runs over all \( N_{nodes} \) nodes in the 2D in-plane finite element mesh. The second-order accurate discretization in space of Eq. (1a)-(1c) is performed in the 2D finite element planes according to Eqs. (4) where \( \nabla = (\partial/\partial z, \partial/\partial r, 0) \), \( \mathbf{u} = (u_z, u_r, 0) \) and \( d\Omega = dr \, dz \):

\[
\int_{\Omega} r \left( N_i R_{\text{cont}} + \tau_{p.pg} \nabla N_i \cdot \mathbf{R}_{\text{mom}} \right) d\Omega = 0 \tag{4a}
\]

\[
\int_{\Omega} r \left( N_i + \tau_{s.pg} \mathbf{u} \cdot \nabla N_i \right) \mathbf{R}_{\text{mom}} d\Omega = 0 \tag{4b}
\]

\[
\int_{\Omega} r N_i R_{\text{charge}} d\Omega = 0 \tag{4c}
\]

The definitions of the stabilization parameters are constant over each triangular element and are inspired from Ref. 27, 28:

\[
\tau_{p.pg} = \left( \frac{1}{\tau_t} + \frac{1}{\tau_{c,p}} + \frac{1}{\tau_{\nu,p}} + \frac{1}{\tau_j} \right)^{-1/2} \quad \tau_{s.pg} = \left( \frac{1}{\tau_t} + \frac{1}{\tau_{c,u}} + \frac{1}{\tau_{\nu,u}} + \frac{1}{\tau_j} \right)^{-1/2} \tag{5a}
\]

where

\[
\tau_t = \frac{\Delta t}{2} \quad \tau_j = \frac{\rho}{\sigma B^2} \tag{5b}
\]

\[
\tau_{c,p} = h_{c,p} \quad \tau_{\nu,p} = \frac{h_{\nu,p}}{4\nu} \quad h_{c,p} = \sqrt{\frac{4\Omega_c}{\pi}} \tag{5c}
\]

\[
\tau_{c,u} = \frac{h_{c,u}}{2|U_c|} \quad \tau_{\nu,u} = \frac{h_{\nu,u}}{4\nu} \quad h_{c,u} = \frac{2|U_c|}{\sum_{j=1}^{3} |U_c \cdot \nabla N_j|} \tag{5d}
\]

with \( \Omega_c \) the area of the element and \( U_c \) the mean planar component of the velocity inside the element. The discretized Eq. (4) can be written in the more compact form described in Eq. (6). For the sake of brevity, the matrices \( M_1, M_2, M_3, L_1, L_2 \) and \( L_3 \) are not detailed in this paper (see Ref. 29 for more details about their hydrodynamic components). The mass matrices \( M_1, M_2 \) and \( M_3 \) are linked to the time-derivative dependent terms. The matrices \( L_1, L_2 \) and \( L_3 \) contain all the linear terms of the Navier-Stokes equations. The right-hand side terms \( C_u, C_p \) and \( C_\phi \) contain the non-linear terms that are treated through the pseudo-spectral approach.

\[
\begin{bmatrix}
M_1 + M_2 \frac{\partial}{\partial \theta} + M_3 \left( \frac{\partial^2}{\partial \theta^2} \right) \frac{\partial}{\partial t} \\
L_1 + L_2 \frac{\partial}{\partial \theta} + L_3 \left( \frac{\partial^2}{\partial \theta^2} \right)
\end{bmatrix}
\begin{bmatrix}
\mathbf{u} \\
\phi
\end{bmatrix}
=
\begin{bmatrix}
C_u \\
C_p \\
C_\phi
\end{bmatrix}
\tag{6}
\]

The discretization in the \( \theta \)-direction is introduced by means of a discrete sum of \( N \) Fourier modes

\[
q(z, r, \theta, t) = \frac{1}{N} \sum_{k=-\frac{N}{2}+1}^{\frac{N}{2}-1} \tilde{q}(z, r, \theta) e^{i k \frac{2\pi}{\theta_{\text{max}}} \theta} \tag{7}
\]

where \( \theta_{\text{max}} \) is the maximal azimuthal coordinate of the domain in the direction of periodicity (\( \theta_{\text{max}} = 2\pi \) for domain with a symmetry of revolution such as in this study). The last step of the spatial discretization consists in applying a Direct Fourier Transform to Eq. (6), which leads to a set of independent systems of equations in Fourier space for the unknowns \( \tilde{q} \):

\[
\begin{bmatrix}
M_1 + M_2 \left( \frac{2\pi}{\theta_{\text{max}}} \right) + M_3 \left( \frac{2\pi}{\theta_{\text{max}}} \right)^2 \frac{\partial}{\partial t} \\
L_1 + L_2 \left( \frac{2\pi}{\theta_{\text{max}}} \right) + L_3 \left( \frac{2\pi}{\theta_{\text{max}}} \right)^2
\end{bmatrix}
\begin{bmatrix}
\tilde{\mathbf{u}}_k \\
\tilde{\phi}_k
\end{bmatrix}
=
\begin{bmatrix}
\tilde{C}_u \\
\tilde{C}_p \\
\tilde{C}_\phi
\end{bmatrix}
\tag{8}
\]
or in a more compact form:

\[
M_k \frac{\partial \hat{q}_k}{\partial t} + L_k \hat{q}_k = -\hat{C} \left( q_{-z+1}, \ldots, q_{z} \right)
\]

(9)

where \( q_p \) is the unknown vector in physical space associated with the \( p \)-th finite element plane. We clearly observe here the decoupling of each matrix system for the Fourier modes. The right-hand side, composed of the non-linear terms, couples all the Fourier modes together. For each mode, the matrices \( M_k \) and \( L_k \) are constant in time but differ for each Fourier mode. Once the linear systems presented in Eq. (9) are assembled, both the real and complex components of the Fourier coefficients \( \hat{q}_k \) need to be solved. As the physical solution is purely real, the following symmetry appears in Fourier space:

\[
\hat{q}_-k = [\hat{q}]_k^*
\]

(10)

so that only the first \( N/2 + 1 \) modes need to be solved, the other ones being evaluated through the symmetry relation of Eq. (10). Furthermore, modes 0 and \( N/2 \) are both purely real. They are combined in a single complex-valued linear system so that the size of every systems is always identical for a load balance point of view and only \( N/2 \) linear systems need to be solved in practice.

To impose boundary conditions on the axis \((r = 0)\) is mathematically not compulsory because this edge is of artificial nature. However, the regularity conditions summarized in table 1 are applied in order to obtain a better quality results. The terms between parentheses are not treated explicitly as they are treated implicitly through an integration by parts with the finite element method described in the previous paragraph. Because of the form of the regularity condition for the radial and azimuthal components of velocity of mode \( k = 1 \), the \( r \)- and \( \theta \)-momentum equations are replaced by the linear combination \([r\text{-momentum}]+[\theta\text{-momentum}]\). The regularity condition \( \hat{u}_{r,1} + I \hat{u}_{\theta,1} = 0 \) replaces the first linear combination for all the modes lying on the axis.

<table>
<thead>
<tr>
<th>Mode</th>
<th>( \hat{u}_{z,k} )</th>
<th>( \hat{u}_{r,k} )</th>
<th>( \hat{u}_{\theta,k} )</th>
<th>( \hat{p}_k )</th>
<th>( \phi_k )</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>( \hat{u}_{z,0} = 0 )</td>
<td>( \hat{u}_{r,0} = 0 )</td>
<td>( \hat{u}_{\theta,0} = 0 )</td>
<td>( \hat{p}_0 )</td>
<td>( \phi_0 )</td>
</tr>
<tr>
<td>1</td>
<td>( \hat{u}_{z,1} = 0 )</td>
<td>( \hat{u}_{r,1} = 0 )</td>
<td>( \hat{u}_{\theta,1} = 0 )</td>
<td>( \hat{p}_1 )</td>
<td>( \phi_1 )</td>
</tr>
<tr>
<td>( k \neq 0,1 )</td>
<td>( \hat{u}_{z,k} = 0 )</td>
<td>( \hat{u}_{r,k} = 0 )</td>
<td>( \hat{u}_{\theta,k} = 0 )</td>
<td>( \hat{p}_k )</td>
<td>( \phi_k )</td>
</tr>
</tbody>
</table>

Table 1. Axis boundary conditions.

### III.B. Temporal integration

At this point, we still need to perform a temporal integration on time-derivative dependent terms in Eq. (9). An implicit second-order accurate Crank-Nicolson scheme is used for the linear terms whereas explicit schemes (either Adams-Bashforth or Runge-Kutta with 1 up to 4 stages) are chosen for the non-linear convective and stabilization terms. The family of temporal integrators available in SFELES allows us to keep the numerical dissipation at a low value while the numerical stability of the convective terms is controlled. Finally, it is found more convenient to solve the variations in time of the unknowns \( \delta \hat{q}_k = \hat{q}_{k}^{n+1} - \hat{q}_k^n \).

The combination of the spatial discretization and temporal integration boils down to the form:

\[
\left( \frac{M_k}{\Delta t} + \frac{L_k}{2} \right) \delta \hat{q}_k = -L_k \hat{q}_k^n - \sum_{s=1}^{S} \beta_s \hat{C}^s \left[ \begin{array}{c} b_k \end{array} \right]
\]

(11)

where \( \beta_s \) and \( S \) depends on the chosen time integrator for the non-linear terms.

### III.C. Parallel implementation

To exploit the advantages of the combined spectral/finite element discretization, three levels of parallelism coexist in SFELES.

- First, the physical domain is partitioned by METIS in as many parts as the total number of processes launched for the computation. Each process assembles its own local matrices \( A_k \) and right-hand side
\( b_k \) for all the required modes. Doing so, the direct and inverse Fourier transforms are performed locally and do not require any communication between processes.

- As the matrices \( A_k \) in Eq. (11) are constant in time, one can factorize each matrix \( A_k \) in LU factors independently from the other matrices. Similarly, as the linear systems associated with each Fourier mode in Eq. (11) are independent from each other within each time step, the substitution phase for one linear system does not require any information from the other systems and can be performed independently.

- The last and most recent level of parallelism is related to the ability to perform the factorization and substitution phases for each mode on several processes thanks to the parallel direct solver MUMPS\(^{31,32} \).

### IV. Numerical results

The numerical results obtained for a hydrodynamic Reynolds number \( Re_b = u_b d / \nu = 8000 \) based on the bulk velocity and pipe diameter and a Hartmann number \( 0 \leq Ha = BR \sqrt{\sigma / \rho \nu} \leq 100 \) are presented in this section. A time-varying forcing term in the z-momentum equation ensures a constant \( Re_b \) during the whole computation. The length of the pipe is set to \( L_z = 5d \). The present computations are performed with \( 250 \times 100 \times 128 \) nodes/modes in respectively the axial, radial and azimuthal directions for a total of 3.2 million nodes. The nodes of the bidimensional structured finite element mesh are uniformly spaced in the axial direction, whereas the mesh is refined near the wall of the pipe in the radial direction. In viscous wall units (\( \nu / u_r \)), the grid spacing is \( \Delta r^+ \approx 0.09 \) at the pipe wall, \( \Delta r^+ \approx 6.6 \) near the centreline of the pipe and \( \Delta z^+ \approx 5.3 \). The circumferential grid spacing is \( (d \Delta \theta / 2)^+ \approx 8.5 \) near the wall. These characteristics of the mesh are quite similar to those of Ref. 33,34. The time integration step is equal to \( \Delta t = 10^{-4} \approx 1.3 \times 10^{-5} t^* \), where \( t^* \) is the dimensionless time scale defined as \( t^* = d / u_r \). The flow is computed until it reaches a fully developed state (until \( t = 10 t^* \)). From \( t = 10 t^* \), the statistics were gathered until \( t = 14 t^* \) and result from a spatial average along the axial direction and a time average over each time step. Each computation required a total CPU-time of 50,000 CPU-hours. Table 2 provides a summary of the flow conditions.

<table>
<thead>
<tr>
<th>Table 2. Mean flow configurations for the numerical simulations.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ha = ( BR \sqrt{\sigma / \rho \nu} )</td>
</tr>
<tr>
<td>( Re_b = u_b d / \nu )</td>
</tr>
<tr>
<td>( Re_{\tau} = u_{\tau} d / \nu )</td>
</tr>
</tbody>
</table>

\(^*\) Hartmann number with \( B \) the applied magnetic field, \( R \) the radius of the pipe, \( \sigma \) the electrical conductivity of the fluid, \( \rho \) the density of the fluid and \( \nu \) the kinematic viscosity of the fluid.

\(^\dagger\) Hydrodynamic Reynolds numbers with \( d \) the diameter of the pipe, \( u_b \) the average velocity and \( u_{\tau} \) the wall shear velocity defined by the wall shear stress \( \tau_{\text{w}} = \rho u_{\tau}^2 \).

### IV.A. Instantaneous solution

The general pattern of the solution is presented in figure 2 with the instantaneous streamwise velocity component \( u_z \) in a cross-section at \( z = L_z / 2 \) and in the meridional \( x - z \) plane. We clearly see that for \( Ha = 0, 5 \) and 10, the regime is turbulent and from \( Ha = 15 \) the flow is entirely laminarized. For the turbulent cases, the core is flattened when the Hartmann number is increased. This laminarization process first occurs in the core and the Hartmann layer as clearly shown in figure 2(h). This laminarization process in the core of the pipe and the Hartmann layers is also highlighted in figure 3, where turbulent structures vanish in the plane parallel to \( B \) as \( Ha \) increases. The Roberts layers are less affected by the increasing Hartmann number than the Hartmann layers. For the considered Reynolds number \( Re_b = 8000 \), we do not obtain the quasi-laminar state observed with a Reynolds number \( Re_b = 10^5 \) in a square duct\(^{24} \). From \( Ha = 15 \), the turbulent structures completely disappear and we retrieve the usual laminar distribution of velocity predicted for low Hartmann numbers and by an asymptotic approach\(^{35,36} \). It is interesting to note that the transition between a partially turbulent and a fully laminar flow is brutal, as a slight increase of the magnetic field between the case \( Ha = 10 \) and \( Ha = 15 \) is sufficient to completely laminarize the flow.
Figure 2. Instantaneous velocity field (a) to (e) in cross-section $x-y$ located at $z = L_z/2$ and (f) to (i) in the meridional $x-z$ plane (parallel to B). All the plots have the same contour level ranging from 0 (blue) to 5.03 (red).
IV.B. Mean flow and turbulent fluctuations

The time and streamwise-averaged $u_z$ velocity profile is shown in figure 4 in the direction parallel and perpendicular to the external magnetic field $B$. An obvious consequence of an increasing Hartmann number is the flattening of the velocity profile in the core of the pipe in the direction parallel to $B$. In this direction, the flow becomes nearly $r$-independent outside the boundary layers. The MHD turbulent profiles ($Ha = 5$ and $Ha = 10$) are slightly less rounded than their hydrodynamic counterpart. This alteration is more visible in the direction parallel to $B$ because of the so-called ‘Hartmann effect’. In the direction perpendicular to the magnetic field, the core velocity slightly increases between $Ha = 0$ and $Ha = 5$, then decreases for $Ha = 10$. As explained in Ref. 24, this behaviour is attributed to the reduction of the wall-normal turbulent transport of momentum. The decreasing core velocity is further observed with the laminar results. This is linked with the flattening of the velocity profile.

The velocity fluctuations are represented by the root-mean-square values $u_{ij,rms} = \overline{u_i u_j} - \overline{u_i} \overline{u_j}$, where $\overline{\ldots}$ is the temporal average operator. Figure 5 shows the distribution of the fluctuations of each velocity component as well as the Reynolds-stress components $u_{zr,rms}$. The results are non-dimensionalized by the friction velocity $u_\tau$. In these cuts, we represent only the turbulent cases because the fluctuations are equal to zero for laminar cases. An obvious observation is the damping of turbulence with an increasing Hartmann number. We can see that this damping is slightly more pronounced in the direction parallel to $B$ than in the direction perpendicular to $B$.
Figure 5. Mean velocity fluctuations in a direction parallel (left) and perpendicular (right) to \( B \). The averages are performed over time and along the streamwise coordinate \( z \).
IV.C. Friction coefficient

The wall shear stress $\tau_w = \rho u_b^2$ defined in table 2 is computed using the global wall-normal derivative of the mean axial velocity. In figure 6, the global skin friction coefficient $C_f = \tau_w / \rho u_b^2$ is represented as a function of the ratio $Ha/Re_b$. For the sake of comparison, we also provide numerical results of Satake and Loeffler & co. In the laminar regime ($2Ha/Re_b \leq 25$) and then to increase linearly in the turbulent regime ($2Ha/Re_b \geq 37.5$). This evolution of the skin friction coefficient is a combination of two effects: the steeper velocity profile in the Hartmann layer in the laminar regime and the damping of turbulence in the turbulent regime. The fact that the flow at $2Ha/Re_b = 37.5$ in Ref. 11 is still in the turbulent regime is maybe an indicator that our result at $Ha = 15$ is not fully reliable. Indeed, as underlined in Ref. 37, the transition point between turbulent and laminar case covers a relatively wide range of $Ha/Re$. Several factors can influence the transition: the periodic length along the streamwise direction, the regime of the initial condition, etc. In the purely hydrodynamic case ($Ha = 0$), the skin friction coefficient was found to be very close to the analytical value obtained by Blasius’ law $C_f = 0.079Re_b^{-1/4}$. In the laminar regime, the slope of the line is slightly larger than the analytical slope of $3\pi/8$ obtained by Shercliff. This discrepancy is also observed for the other numerical/experimental values but no satisfactory explanation has been found so far.

IV.D. Physical and numerical dissipations

One main advantage of the finite element method is the ability to compute a posteriori the energy budget given by Eq. (12a). The variation of the kinetic energy $e$ is compensated by the viscous and Joule dissipations ($\epsilon_\nu$ and $\epsilon_J$), by the introduction of energy through the source term $\epsilon_f$ that maintains a constant bulk Reynolds number $Re_b$ and the contribution of the PSPG and SUPG stabilization terms $\epsilon_{stab}$.

$$\frac{\partial e}{\partial t} = -\epsilon_\nu - \epsilon_J + \epsilon_f - \epsilon_{stab} \quad (12a)$$

where

$$e = \frac{1}{V} \int_V \frac{u \cdot u}{2} \, dV \quad (12b)$$

$$\epsilon_\nu = \frac{1}{V} \int_V \nu \nabla u : \nabla u \, dV \geq 0 \quad (12c)$$

$$\epsilon_J = \frac{1}{V} \int_V \frac{J \cdot J}{\rho \sigma} \, dV \geq 0 \quad (12d)$$

$$\epsilon_f = \frac{1}{V} \int_V u \cdot f \, dV \quad (12e)$$

All these contributions are shown in figure 7. Concerning the viscous and Joule dissipations, we notice the appearance of peaks of dissipation in the Hartmann layers. These peaks are more important for high Hartmann numbers because of the narrowing Hartmann layer. The narrowing Hartmann layer causes steeper velocity gradients and narrower passages for the electric current. In contrast, the evolution of the dissipations is smooth in the Roberts layers. Concerning the contributions of the stabilization terms in the energy budget, we see that their amplitude is 2 up to 3 orders of magnitude lower than the physical dissipations. The additional numerical stabilization terms do not alter the physics of the solution and allow a smooth evolution of the solution without any spurious oscillations.
Figure 7. Mean physical and numerical dissipations in a direction parallel (left) and perpendicular (right) to \( B \). The averages are performed over time and along the streamwise coordinate \( z \). The numerical dissipations are shown only in the direction parallel to \( B \).
V. Conclusion

The present work consisted of a numerical study of isothermal, turbulent MHD pipe flow in a transverse, uniform and constant magnetic field. A numerical algorithm was developed for this purpose. At low Reynolds number, we did not observe the particular pattern of quasi-laminar flow observed at higher Reynolds number in a square duct.\textsuperscript{24} The transition between turbulent and laminar regimes was found to occur around $Re/2Ha \sim 250$, which is in good accordance with previous works\textsuperscript{7,11}.

Velocity profiles in the direction parallel to the magnetic field flatten as the Hartmann number increases. Along the direction perpendicular to the magnetic field, the profiles became more rounded than the zero-field profile. We observed that the laminarization process in the turbulent regime starts first in the core and the Hartmann layers. This observation was made on the basis of the velocity fluctuations.

The skin friction coefficient has been found to be in good agreement with the literature. As the Hartmann number increases, the friction tends to decrease in the turbulent regime and then to increase linearly in the laminar regime.

The viscous and Joule dissipations show why the laminarization process occurs first in the Hartmann layers and the core of the pipe. The combination of the two energy-dissipating processes is more pronounced in these zones. It was also proved that the contribution of the stabilization terms in the numerical scheme is negligible with respect to the viscous and Joule dissipations and does not alter the physics of the flow.

Acknowledgments

This work was supported by a grant from the Belgian Fonds de la Recherche Scientifique de Belgique (FRS-FNRS). This work utilized the Janus supercomputer, which is supported by the National Science Foundation (award number CNS-0821794) and the University of Colorado Boulder. Computational resources have also been provided by the supercomputing facilities of the Université Catholique de Louvain (CISM/UCL) and the Consortium des Équipements de Calcul Intensif en Fédération Wallonie Bruxelles (CECI) funded by the Fonds de la Recherche Scientifique de Belgique (FRS-FNRS).

References


