COMSOL as a tool for studying Magneto-Hydro-Dynamic effects in liquid metal flow under transverse magnetic field

S. Sahu, R.P.Bhattachryay
E. Rajendrakumar
IPR
Content

- Introduction
- Problem definition
- Equations involved
- Mesh
- Results
- Conclusion
Introduction

✓ Magneto-hydro-dynamics (MHD) is the study of dynamics of an electrically conducting fluids under the presence of transverse magnetic field
✓ The fundamental concept behind MHD is that magnetic fields can induce currents in a moving conductive fluid, which in turn creates forces on the fluid and also changes the magnetic field itself.
✓ The field of MHD was initiated by Hannes Alfvén and got Nobel prize in 1970
✓ Examples of such fluids: plasmas, liquid metals, and salt water or electrolytes
✓ MHD applications: Astrophysics (planetary magnetic field), MHD pumps (1907), MHD generators (1923), MHD flow meters (1935), MHD flow control (reduction of turbulent drag), Magnetic filtration and separation, Fusion reactors (blanket, divertor, limiter, FW)
✓ To study issues related to aforesaid examples using a software it is highly essential to validate the software
✓ At present, It was tried to validate COMSOL as a tool for studying the MHD issues in liquid metal flows
✓ A well established relation [1]J.C.R.Hunt (1965) was used and the same was studied in COMSOL to have a comparison
Problem Definition

Ininitely conducting Hartmann wall

- 25×25 mm² square channel
- 1mm thick wall
- Uniform magnetic field 1T~4T
- Hartmann number (Ha) = 250 ~ 1038
- Steady state Velocity profile is studied

\[ Ha = B d \sqrt{\frac{\sigma}{\rho \nu}} \]

What to expect?

MHD profile

Hydro-dynamic profile

Hartmann number is the ratio of electromagnetic force to the viscous force

Hartmann wall is the wall perpendicular to the magnetic field

Side wall is the wall parallel to the magnetic field.

[J.C.R. Hunt (1965)]
Equations Involved

**Assumptions**

- Steady state laminar flow
- Incompressible Newtonian fluid
- No slip at the wall-liquid interface
- Quasi-static approximation

Navier Stoke’s equation (Momentum Conservation)
\[
\rho (\mathbf{u} \cdot \nabla) \mathbf{u} = \nabla \left[ \rho \mathbf{I} + \mu (\nabla \mathbf{u} + (\nabla \mathbf{u})^T) \right] + \mathbf{J} \times \mathbf{B}
\]

Continuity equation (Mass Conservation)
\[
\rho \nabla \cdot \mathbf{u} = 0
\]

Generalised Ohm’s Law
\[
\mathbf{J} = \sigma (\mathbf{E} + \mathbf{u} \times \mathbf{B})
\]

Current conservation Equation
\[
\nabla \cdot \mathbf{J} = 0
\]

Maxwell’s equation
\[
\mathbf{E} = -\nabla \varphi
\]

Poisson’s Equation
\[
\nabla^2 \varphi = \nabla \cdot (\mathbf{u} \times \mathbf{B})
\]

**Boundary Conditions**

- For Fluid flow
  - Inlet \( u_x = 0.01 \text{ m/s} \), \( u_y = u_z = 0 \)
  - Outlet P=0

- For Electromagnetic analysis
  - \( \varphi = 0 \) at all boundaries
  - \( \mathbf{n} \cdot \mathbf{J} = 0 \) and \( \mathbf{n} \times \mathbf{A} = 0 \) at all outer boundaries
Peculiarity about Liquid Metal MHD

- A number of equation to be solved simultaneously
- Normal Pipe flow gets modified into M-shaped profile (Instable flow)
- Solver settings has to be set properly
- Hartmann layers thickness (\(\sim 1/\text{Ha}\))
  
  \[
  \delta = \frac{1}{B} \sqrt{\frac{\rho U}{\sigma}}
  \]

- Side layer thickness (\(\sim 1/(\text{Ha})^{1/2}\))
  
  \[\sim 6.2\times 10^{-2} - 3.1\times 10^{-2} \text{ in our case} \]

- Mesh should be sufficiently dense to capture the phenomena occurring at this thickness
Mesh

Minimum Mesh thickness along Hartmann layers = 2e-6 m
Minimum Mesh thickness along side layers = 2e-4 m

Inlet will have more velocity variation as compared to outlet

Hartmann layer thickness \( \sim 1/\text{Ha} \)
Side layer thickness \( \sim 1/(\text{Ha})^{1/2} \)
### Velocity Comparison

<table>
<thead>
<tr>
<th>Ha</th>
<th>Hunt</th>
<th>COMSOL</th>
<th>Peak Velocity</th>
<th>Dip -ve Velocity</th>
<th>Core velocity</th>
</tr>
</thead>
<tbody>
<tr>
<td>260</td>
<td>0.09163</td>
<td>0.090084</td>
<td>0.0004762</td>
<td>0.003429</td>
<td></td>
</tr>
<tr>
<td>519</td>
<td>0.1336</td>
<td>0.1314</td>
<td>-0.002668</td>
<td>0.003142</td>
<td></td>
</tr>
<tr>
<td>780</td>
<td>0.1663</td>
<td>0.1633</td>
<td>-0.004383</td>
<td>0.003039</td>
<td></td>
</tr>
<tr>
<td>1038</td>
<td>0.1938</td>
<td>0.1868</td>
<td>-0.005673</td>
<td>0.002966</td>
<td></td>
</tr>
</tbody>
</table>
Velocity Comparison

<table>
<thead>
<tr>
<th></th>
<th>Ha = 519</th>
<th>Ha = 780</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hunt</td>
<td>0.003133</td>
<td>0.003024</td>
</tr>
<tr>
<td>COMSOL</td>
<td>0.003179</td>
<td>0.003033</td>
</tr>
</tbody>
</table>

Core velocity

Ha = 519

Ha = 780
Pressure Comparison

<table>
<thead>
<tr>
<th></th>
<th>$H_a = 260$</th>
<th>$H_a = 519$</th>
<th>$H_a = 780$</th>
<th>$H_a = 1038$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hunt</td>
<td>2.66e3</td>
<td>9.73e3</td>
<td>2.11e4</td>
<td>3.63e4</td>
</tr>
<tr>
<td>COMSOL</td>
<td>2.7e3</td>
<td>9.85e3</td>
<td>2.12e4</td>
<td>3.67e4</td>
</tr>
</tbody>
</table>
**Conclusion**

- A model was simulated using COMSOL for a well known analytical relation.
- The COMSOL results are matching well (Max 4% error) with the analytical results for velocity and pressure measurements, except at dip –ve velocity locations (Max 25% error).
- It is expected that the error can be minimised by increasing mesh density at that locations.
Thank You
Reference

- “Magnetofluiddynamics in Channel and containers”, U. Muller, L. Buhler, Springer.
- “Magnetohydrodynamics”, R. MOREAU, Vol3, kluwer academic publishers
- User’s Manual, COMSOL 4.3b