Simulation of Incompressible magneto-hydrodynamic flows (MHD) using mhdFoam solver.

Abstract.

This report aims to simulate and predict the 2-D magneto-hydrodynamic flows between two infinite parallel, placed/ oriented perpendicular to the direction of the magnetic field, using the OpenFOAM software. This report also aims to compare the velocity distribution with analytical results. MHD flows have been a topic of interest for many research activities and industrial applications including nuclear fusionreactors, materials engineering and metallurgy, therefore this report will decribe a theoretical background and its implementation using OpenFOAM solvers.

Problem Statement.

The flow being simulated is of an electrically conducting fluid, driven by a constant pressure gradient between two insulated plates which are orthogonal to the magnetic field direction. This type of flow is popularly called as a *Hartmann flow* (Named after the scientist who performed this experiment). The mesh has been generated/ developed using the OpenFOAM's *blockmesh* utility, and simulations have been done using the *mhdFoam* solver.

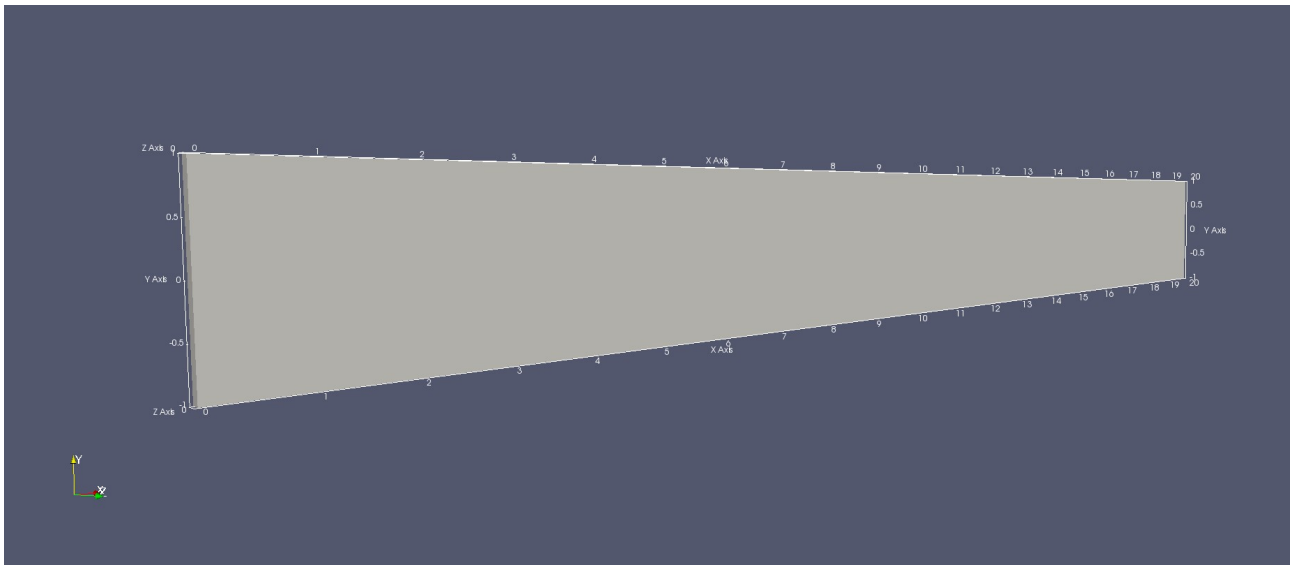

Fig.1. Dimensions and orientation of the geometry.

Free stream velocity: 1m/s
Hartmann Numbers: 10, 50, 100 and 500.
Hydraulic diameter : 1m
Magnetic Reynold's number = 1.