

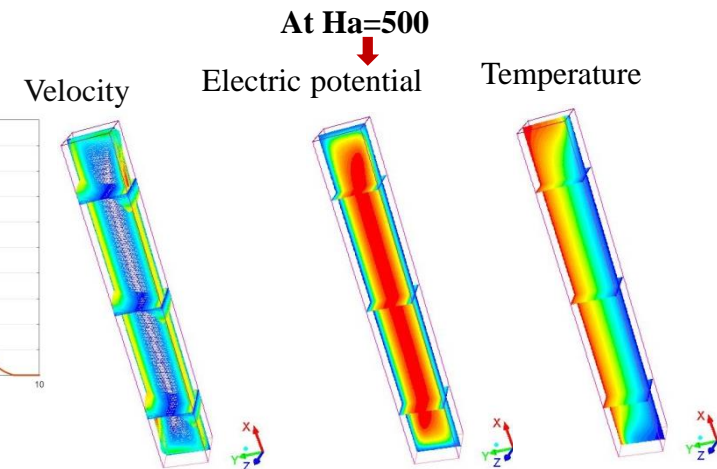
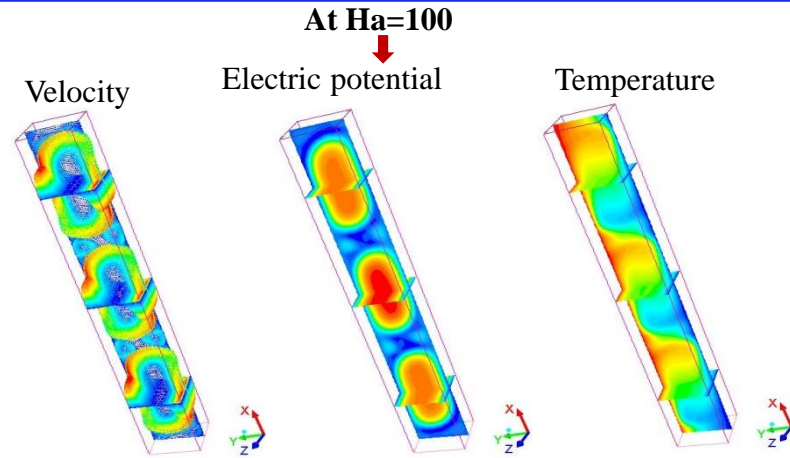
APPLICATION OF ANSYS FLUENT MHD CODE FOR LIQUID METAL MHD STUDIES

A. PATEL and R. BHATTACHARYAY

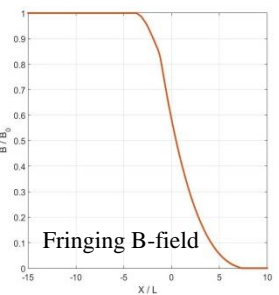
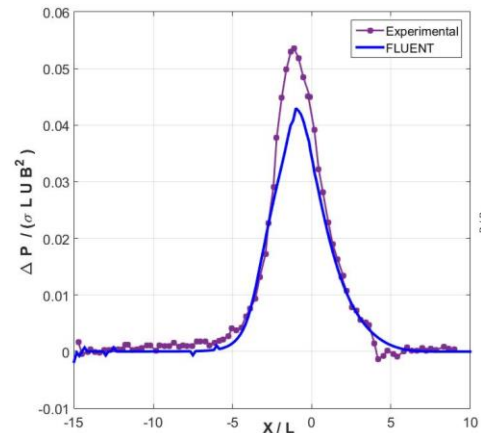
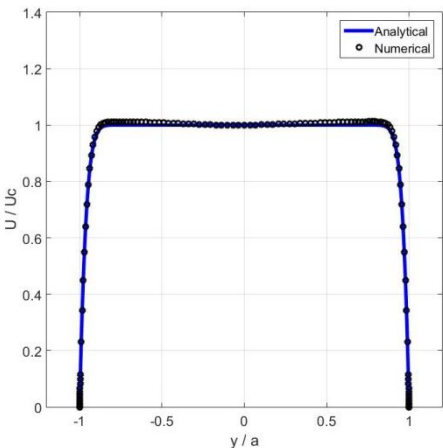
Institute for Plasma Research

Ahmedabad, India

- MHD studies are required to assess the performance of liquid breeder blankets.
- The ANSYS FLUENT is being used for performing MHD studies. In the present work, three MHD benchmark problems have been analysed.
 1. 2D fully developed laminar steady MHD flow for non-conducting square duct at $Ha = 500$.
 2. 3D laminar, steady developing MHD flow in a square conducting duct under the fringing magnetic field at $Ha=2900$, $N= 540$.
 3. Natural convection transient MHD flow inside vertical square enclosure at $Ha = 100$ for $Gr= 4 \times 10^6$, $Pr=0.025$, (here $\nabla T \perp \mathbf{r} \cdot \mathbf{B}$).
- In the future, the 3D Thermo-fluid MHD analysis of the liquid breeder blanket are planned using FLUENT.



Natural Convection: The FLUENT results are found to be in good agreement with the results reported by G. Authié and T. Tagawa, et al. Three main vortices have been observed at lower magnetic field ($Ha=100$). The magnetic field has been found to be the controlling parameter for the natural convection.



FLUENT and experimental results for the variation of the transverse pressure difference at $Ha=500$, $N= 540$

3D MHD effects: The results for the spatial variation of normalised transverse pressure difference from FLEUNT agrees well with the experimental data.

Numerical and analytical results for the normalised velocity profile across the two side wall at $Ha=500$. The numerical results for velocity profile are found to be in good agreement with those from analytical solution.