Contribution ID: **552** Type: **Poster**

Application of ANSYS FLUENT MHD Code for Liquid Metal MHD Studies

Wednesday, October 24, 2018 8:30 AM (4 hours)

Magneto Hydro Dynamic (MHD) phenomena plays an important role in governing liquid metal flow characteristics under strong transverse magnetic field and has, therefore, gained the attention of fusion community for the design of liquid breeder blankets. In presence of plasma confining toroidal magnetic field, the flow of electrically conducting liquid metal (Li/Pb-Li), typically used for coolant and/or tritium carrier, is greatly affected due to flow opposing Lorentz force, which arises due to interaction between magnetic field and induced current in the liquid metal. For the successful design and development of liquid breeder blankets, detailed MHD analysis is highly desirable to understand various effects of MHD, such as change in velocity profile, pressure drop, heat transfer etc.
The liquid metal MHD studies are being carried out using both analytical and numerical approaches. The analytical solutions, derived under 2D fully developed flow approximations, are limited to the simple flow geometries and hence they are not applicable for the analysis of complex blanket flow configuration, which consists of bends, transition zone, multichannel flow etc. Numerical simulation techniques are, therefore, used extensively to perform MHD analysis in such complex flow configuration and various MHD codes, either newly developed or commercially available are being reported. The MHD code, however, needs to be benchmarked extensively and validated before its application to complex flow configuration in liquid breeder blanket. In the present work, three MHD benchmark problems of ref. [1] has been successfully analyzed using ANSYS FLUENT MHD code and results are compared with available literature data. The selected problems are (i) 2D fully developed laminar steady MHD flow, (ii) 3D laminar, steady developing MHD flow in a non-uniform magnetic field and (iii) MHD flow with heat transfer (buoyant convection). The results have provided more confidence in using FLUENT as a promising MHD analysis tool for fusion application. The numerical model, analysis, methodology and simulation results of each benchmark problem will be discussed in detail.

References:

1. S. Smolentsev, S. Badia , R. Bhattacharyay etal, FED 100 (2015) 65–72.

Country or International Organization

India

Paper Number

FIP/P3-30

Primary author: Mrs PATEL, Anita (Institute For Plasma Research)

Co-author: Dr BHATTACHARYAY, RajendraPrasad (Institute For Plasma Research)

Presenter: Mrs PATEL, Anita (Institute For Plasma Research)

Session Classification: P3 Posters

Track Classification: FIP - Fusion Engineering, Integration and Power Plant Design