Fusion Engineering and Design 127 (2018) 151–159

Contents lists available at ScienceDirect

Fusion Engineering and Design

journal homepage: www.elsevier.com/locate/fusengdes

Validation of COMSOL code for analyzing liquid metal magnetohydrodynamic flow

S. Sahu*, R. Bhattacharyay

Institute for Plasma Research, Bhat, Gandhinagar, Gujarat, 382428, India

ARTICLE INFO

Keywords:
Fusion reactor
Hartmann number
Interaction parameter
Liquid breeder blanket
Magnetohydrodynamics
Wall conductance ratio

ABSTRACT

Applicability of COMSOL multiphysics software for analyzing liquid metal MagnetoHydroDynamic (MHD) effect under fusion relevant parameters has been verified. For this purpose, few benchmark problems recommended by fusion MHD community have been simulated. The selected benchmark problems are steady state fully developed MHD duct flow, duct flow under transverse fringed magnetic field and transient phenomenon of natural convection in presence of magnetic field. Apart, from the recommended benchmark problems, an experimental scenario has been simulated having flow geometry with multiple 90° bends. For these simulations, two to three available independent physics modules have been suitably coupled under the single platform of COMSOL. The computed numerical results show good agreement with the available analytical or experimental or reported numerical data in the range of MHD parameters Hartmann number $\sim 10^4$, Interaction parameter $\sim 10^4$, wall conductance ratio $\sim 10^{-2}$ & Grasshof number $\sim 10^6$.

1. Introduction

Liquid Breeder Blanket (LBB) concepts are very attractive option for fusion power reactor due to their potentially high thermal efficiency and high tritium breeding ratio [1–5]. Within such LBB, liquid Li or Pb-Li serves as the high grade heat extractor and tritium breeder. Being, electrically conducting, the flow of aforesaid liquids will exhibit strong MHD effects under fusion reactor's magnetic field. As a result, liquid's velocity profile changes significantly, impacting the associated temperature and pressure distribution. This in turn, profoundly affects the blanket performance. Therefore, a better understanding of MHD is necessary for designing a LBB. Numerous asymptotic theories have been investigated in past several decades on MHD duct flows [6–9]. Nevertheless, their applicability is limited up to simple straight channel duct flows and are far away from predicting the overall MHD effect envisaged inside the complex geometry of LBB. Under these circumstances, numerical simulation seems to be a favorable option. MHD computations were pioneered in 1970′s. Yet, until 1980′s MHD simulations were limited up-to few hundreds of Hartmann number ($\mathrm{Ha} = aB\sqrt{\sigma/\mu}$, Ha^2 represents the ratio of electromagnetic forces to viscous forces). This may be because of special numerical difficulty faced under high Hartmann number computation, which is, to numerically resolve the Hartmann and Shercliff layers accurately, that scales $\sim a/\mathrm{Ha}$ and $\sim a/\mathrm{Ha}^{1/2}$ respectively. On the other hand, with the advancement of computer technology, reports of MHD duct flows at fusion relevant Hartmann Number (Ha $\sim 10^4$–10^5) have emerged from the beginning of 21st century.

A number of 3D MHD codes are prevalent among fusion researchers, which can be classified into three categories, viz. (1) Homemade codes, (2) Readymade codes and (3) semi-Homemade codes. Here, Homemade codes refer to those where researchers have indigenously developed the code. HIMAG, MTC and FEMPAR are examples of such Homemade codes [10–12]. Readymade codes refer to those, where, MHD packages are deliberately provided by the supplier. To name a few, FLUENT, CFX & FLUIDYN are such type of Readymade codes [13–15]. Whereas, semi-Homemade codes are those, where, manufacturers do not provide dedicated MHD package, however, researchers have suitably coupled/modified the available packages for solving MHD problems. OpenFoam is an example of such type of semi-Homemade code [16,17]. In the present work, an attempt has been made towards validating a semi-Homemade type COMSOL code for analyzing liquid metal (LM) MHD problems. As such, there is no in-built MHD package in COMSOL; however, a suitable coupling strategy has been employed among few available packages. The major motivation behind this work is to foresee COMSOL as a tool for studying combined multi-physics phenomena associated with LBBs. COMSOL provides a single platform where, a number of physics based modules are available independently, viz. electromagnetism (AC/DC module), fluid dynamics (CFD module), heat transfer (Heat Transfer module), mass transport (Chemical Reaction Engineering Module), user defined differential equations (PDE module)

* Corresponding author.
E-mail address: srikanta@ipr.res.in (S. Sahu).

https://doi.org/10.1016/j.fusengdes.2018.01.009
Received 6 March 2017; Received in revised form 7 November 2017; Accepted 2 January 2018

Nomenclature

Non-dimensional numbers

C_w	Wall conductance ratio
Gr	Grassh of number
Ha	Hartmann number
N	Interaction parameter
Pr	Prandtl number
$\tilde{Q}$	Flowrate

Symbols

a	Characteristics length equal to half width of flow dimension parallel to the magnetic field
b	Width of the flow dimension perpendicular to magnetic field
$\boldsymbol{B}$	Magnetic field vector
B	Magnitude of magnetic field
$B_{\max}$	Maximum value of magnetic field
$\boldsymbol{g}$	Acceleration due to gravity
$\boldsymbol{J}$	Electric current vector
$\boldsymbol{J}_{x,y,z}$	Electric current vector along x,y,z axes
$\boldsymbol{J}_{zw}$	Z-component of wall electric current
L	Length of the duct
p	Dynamic pressure
Q	Flowrate
t	Time
T	Temperature
T_0	Reference temperature
t_w	Wall thickness
u	Mean velocity
$\boldsymbol{u}$	Velocity vector
u_c	Duct core velocity
$u_{x,y,z}$	Velocity components along x,y,z axes
β	Thermal expansion coefficient
k	Thermal conductivity
ρ	Density
ν	Kinetic viscosity
σ	Electric conductivity
σ_w	Electrical conductivity of wall
φ	Electric potential

etc [18]. As of now, PDE module is being used by Ying et al. for studying tritium transport in LBB system [19]. The current work is a coupling of AC/DC, CFD and Heat Transfer modules of COMSOL to compute LM thermo-fluid MHD phenomena. In future, more modules can be combined with the above-mentioned modules for analyzing MHD flow together with its impact on H^3 transport, corrosion of structural material and thermal stresses etc. provided, the adopted coupling strategy/code are validated for each individual phenomenon against available experimental/analytical/numerical data.

Smolentsev et al. has proposed a set of benchmark problems for the validation of LM MHD codes [20]. From the proposed benchmark problems, most of the available MHD codes have been validated against the 2D fully developed MHD flow under uniform magnetic field [10–17]. Few codes like HIMAG, OpenFoam, and MTC have also been verified for 3D MHD flows under non-uniform magnetic field [21–24]. Only two codes, HIMAG & OpenFoam have been used for MHD flows with heat transfer [25,26]. Apart from this, HIMAG has also been used for many other problems pertaining to DCLL type LBB design [27,28]. Other codes have been used for many other steady state isothermal MHD problems [29–31].

In the present work, the COMSOL code validation has been performed by computing three benchmark problems suggested in [20]. The selected benchmark problems comprise of scenarios with steady state MHD flow under uniform as well as non-uniform magnetic field and the transient MHD flow with heat transfer. More details about the computed problems have been provided in Section 3, accompanied by, the comparison plots of computed physical parameters against their available analytical or experimental or reported numerical values. The relevant numerical methodology and coupling strategy have been described in Section 2. Finally, the conclusion and future scope of the work is discussed in Section 4.

2. Numerical methodology

In general, liquid metals behave as a Newtonian fluid and can be treated as incompressible. Moreover, the flow conditions inside fusion reactor will have low magnetic Reynolds number $<<1$, hence, can also be considered to be induction less. Taking into account these flow behaviors, the numerical strategy employed is based upon the computation of electric potential as the main electromagnetic variable and to use it for estimating induced electric currents in the liquid metal and duct and then to couple these currents with the fluid flow. The governing equations can be expressed as follows for the steady state isothermal MHD problems:

$$(\boldsymbol{u}.\nabla)\boldsymbol{u} = -\frac{1}{\rho}\nabla p + \nu\nabla^2\boldsymbol{u} + \frac{1}{\rho}\boldsymbol{J}\times\boldsymbol{B} \tag{1}$$

$$\nabla.\boldsymbol{u} = 0 \tag{2}$$

$$\boldsymbol{J} = \sigma(-\nabla\varphi + \boldsymbol{u}\times\boldsymbol{B}) \tag{3}$$

$$\nabla.\boldsymbol{J} = 0 \tag{4}$$

Eqs. (3) and (4) can be combined to form a Poisson type equation

$$\nabla.(\nabla\phi) = \nabla.(\boldsymbol{u}\times\boldsymbol{B}) \tag{5}$$

Where, $\boldsymbol{u}$, p, $\boldsymbol{J}$, $\boldsymbol{B}$ and φ are main variables representing fluid velocity, pressure, electric current density, magnetic field and electric potential respectively. ρ, ν and σ are fluid parameters density, kinematic viscosity and electrical conductivity respectively. Eqs. (1) (2) and (4) represents the momentum, mass and current conservation respectively and Eq. (3) is the generalized Ohm's law.

However, for non-isothermal problems the set of Eqs. (2)–(4) are solved along with the modified version of Eq. (1) and energy balance equation, as given below:

$$\frac{\partial\boldsymbol{u}}{\partial t} + (\boldsymbol{u}.\nabla)\boldsymbol{u} = -\frac{1}{\rho}\nabla p + \nu\nabla^2\boldsymbol{u} + \frac{1}{\rho}\boldsymbol{J}\times\boldsymbol{B} + \beta(T-T_0)\boldsymbol{g} \tag{6}$$

$$\frac{\partial T}{\partial t} + (\boldsymbol{u}.\nabla)T = k\nabla^2 T \tag{7}$$

Where, t, T denotes time and temperature respectively, β is the thermal expansion coefficient of the liquid, T_0 is the reference temperature of the fluid and $\boldsymbol{g}$ is the acceleration due to gravity respectively. k is the thermal conductivity of the material. Eq. (6) accounts for the density change with temperature and portrays Boussinesq approximation. Eq. (7) captures the thermal diffusion inside the liquid and is a consequence of energy balance.

The governing equations show that the velocity vector computed in Eq. (1) is required for Eq. (5) & the electric current density deduced from electric potential solved in Eq. (5) is required in Eq. (1). In other words, Navier-Stokes equation (Eq. (1)) is coupled with the electric potential Poisson type equation (Eq. (5)). For solving above coupled equations in COMSOL, *electric current interface* of AC/DC module (which computes electric potential), is coupled with the velocity components