A Magnetohydrodynamic study af a inductive MHD generator

Augusto Montisci, Roberto Pintus
University of Cagliari, Department of Electrical and Electronic Engineering,
Piazza d'Armi , 1 – 09123 Cagliari – Italy
E-mail: {amontisci,roberto.pintus}@diee.unica.it

Introduction

In this work, a preliminary study has been done to evaluate the physical feasibility of an inductive MHD generator. The MHD Generator has been deeply studied since early of past century [1], as an alternative to the gas turbine based on energy conversion. The aim of the proposed device is to overcome the typical drawbacks of MHD generators, such as the needed of a very high external magnetic field and as a consequence the use of the superconductive coils, the strong dependency of the efficiency by the temperature of the fluid, which affect the degree of conversion of the enthalpy, the deterioration of the electrodes that are in contact with the high temperature plasma. In fact the proposed device does not need an external magnetic fluid to work, but it performs the energy conversion by means of the induction principle. This is possible thanks to a pulsed ionization of the fluid current, carried out with an electrode dipped in the fluid current, followed by a charge separation with a stationary electric field. The aim of this study is to preliminary investigate the physics involved in this process, and to establish if a meaningful level of energy can be transferred to the electric load. In this study the fluidodynamic analysis has been performed, by assuming a multi-physics approach, so that both fluidodynamic and electromagnetic aspects have been treated simultaneously.

Physical Phenomena Description

In the Fig. 1 a 3D image assembly of the proposed generator is represented. A high temperature gas current flows in a duct, where an electrode periodically generates an electric discharge. A stationary external electric field causes a charges distribution dissymmetry, after that the flow is split in two currents, one having an excess of positive charge, the other one negative charge. The electric current along the separated flows is time varying, then each of them induces an electromotive force in a toroidal coils that is wrapped around a core of very high magnetic permeability placed of the cavity and around of the ducts. After that, the two fluid currents are mixed in the terminal duct, where the charges are neutralized.
A sufficient number of charges has to be generated, in order to obtain a meaningful electrical current. A pulsed voltage source periodically generates an electrical discharge in the gas.
The applied physical model to study the problem can be divided into three main modules:

1.electrostatic stationary simulation model to evaluate electrostatic field;
2.fluid flow simulation model based on the Navier-Stokes equations (NS);
3.convection and diffusion simulation transient model to evaluate charge diffusion.

The electrohydrodinamic flow is described by the following equations. Modeling of static electric field is carried out using the electric potential V governed by Poisson equation:

$$\nabla^2 V = \frac{\rho}{\varepsilon_0} \qquad (1)$$

where ρ is the space charge density and ε_0 is the dielectric permittivity of free space. The electric potential is defined from electric field intensity E as:

$$E = -\nabla V \qquad (2)$$

The fluid dynamic part of the problem is described by the Navier-Stokes equation for steady state turbulent gas flow:

$$\rho u \frac{\partial u}{\partial x} + \rho v \frac{\partial u}{\partial y} = \left(\mu + \mu_t\right)\left[\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right] - \frac{\partial p}{\partial x} + F_x \qquad (3)$$

$$\rho u \frac{\partial v}{\partial x} + \rho v \frac{\partial v}{\partial y} = \left(\mu + \mu_t\right)\left[\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right] - \frac{\partial p}{\partial y} + F_y \qquad (4)$$

the turbulent viscosity has been defined through the $\kappa - \varepsilon$ module:

$$\mu_t = C_\mu \rho \frac{\kappa^2}{\varepsilon} \qquad (5)$$

where the constant C_μ has bee set at 0.09 and other constant involved in the computation has been set at the usual value applied for this type of problem (Table 1).

Constants	**Value**
C_μ	0.09
C_1	0.144
C_2	0.192
σ_κ	1.0
σ_ε	1.3

Table 1: Coefficient applied in the $k-\varepsilon$ model to performer the computation

The charge transport by convection and diffusion is described by Fick's law. The convection and diffusion equations describe physical phenomena where the charge positive and negative are transferred inside a physical system due to two process: diffusion and convection. The equation take the form:

$$\frac{\partial c}{\partial t} = D \nabla^2 c R - \vec{v} \cdot \nabla c \qquad (6)$$

where *c* is the concentration of charge, *D* denote the diffusion coefficient, *v* is the velocity vector and R is the reaction rate. The electric current, created by the motion of electric charge, generates a magnetic field. The magnetic flux variation induces an electromotive force in the secondary winding. The number of coils is selected in such a way that the last one gives a meaningful contribution to the overall voltage.

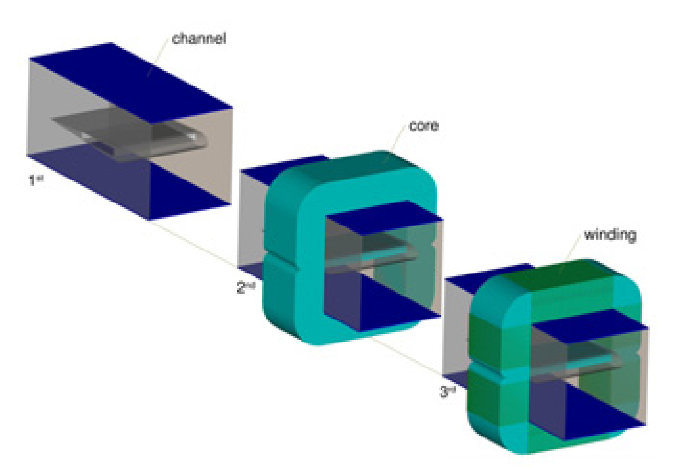

Figure 1: 3D image of the proposed MHD Generator

Numerical Model

Model geometry under study. Fig.1 Shows the chassis of the studied device. This is a duct having squared section, having in the inner part a diaphragm that splits the fluid current in two parts. This diaphragm has a cavity where by means of a very high permeability material is used to realize the core. Around said core the secondary coil is wrapped. In the initial part of the duct, an electrode is placed that generate a pulsed discharge in the fluid current, so producing the charge carries that give rise to the electrical current. The volume under control has been divided into the eight sub-domains shown in fig 2. All domains have been activated in the electrostatic model, while for the fluid flow model and convection and diffusion model were excluded five sub-domains that describe the profile.

The same figure shows the mesh of about 91000 triangular elements, which is more fine close to the discharge region and to the walls. The electric discharge is represented by the smallest of the two concentric circles. The one larger concentric circle were used to define mesh refinement but do not represent actual boundaries.

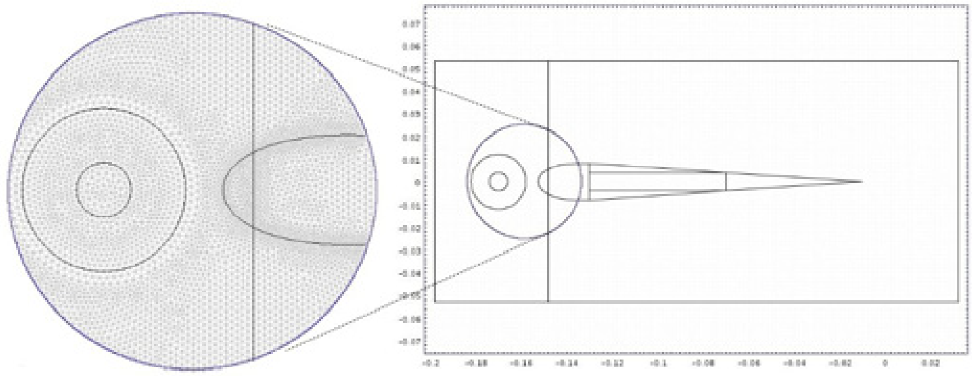

Figure 2: Numerical simulation domain and mesh of generator geometry

Material Properties. The duct is fed with a gas, which derives from a combustion chamber or a gasifier [2] [3]. The temperature of the fluid is set to 1000 K while the density and the dynamic viscosity are a function of temperature and pressure. On the basis of the experimental data retrieved from the literature, it can be assumed that the charge carries concentration, due to the discharge, is about 10^{14} n_e/cm^3 [4]. The total number of charges is assumed to be about $2\cdot10^{15}$, i.e., we assumed that 10 cm^3 are involved in the discharge process. The charge of the same sign is then $n_e\cdot e$=1.6 10^{-4} C. The generation of the charges by means of the electrode is modeled injecting a time-varying flow of positive and negative charge carries. In the volume of the electric discharge a space charge volume density is applied, to the volume of the electrical discharge, by setting the Reaction rate parameter to the following trend:

$$R=\frac{C_R}{\sqrt{2\pi\cdot\sigma^2}}\left[e^{\frac{(t-t_1)^2}{2\cdot\sigma^2}}+e^{\frac{(t-t_2)^2}{2\cdot\sigma^2}}\right] \tag{7}$$

where:
- C_R is the maximum charges concentration in the time
- σ is the parameter that gives the speed of charge diffusion
- t_1 and t_2 are the time instants when two consecutive discharges occur.

The time interval between t_1 and t_2 are stated in such a way the interference between two consecutive discharges is negligible. The Fig. 3 shows the time variation of *R*.

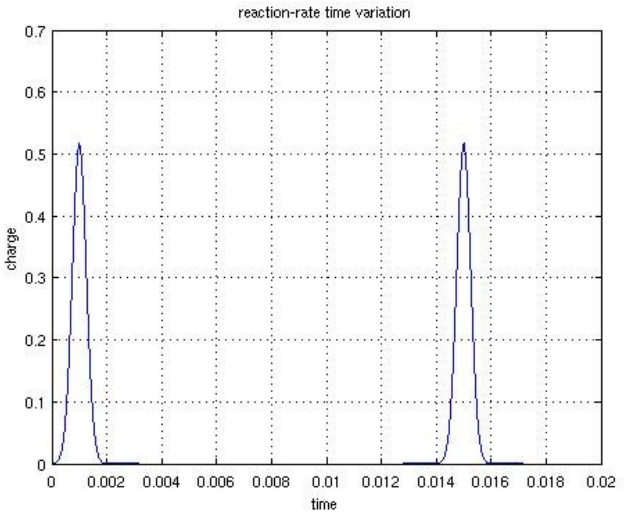

Figure 3: Time variation of R

The charges repulsion effect has been taken into account by increasing the value of the diffusion coefficient obtained by comparing the peripheral diffusion velocity with the velocity of the same particles due to the interaction with the charge distribution. Therefore, the coefficient that without charges interaction would be μ =$5\cdot10^{-5}\text{m}^2/\text{s}$ has been corrected with μ =$2.1208\cdot10^{-4}$ m^2/s. The relative dielectric permittivity of the fluid current is assumed equal to 1.0.

In order to take into account the effect of the external field on the charge carriers, in the subdomain settings of the convection and diffusion model, a velocity term is added to the drift velocity, which depends on the external electrical field and on the mobility of charges. On the basis of the experimental data available in literature, a mobility of $1.8 \cdot 10^{-4}$ $m^2V^{-1}s^{-1}$ has been assumed [5].

Boundary Conditions. For fluid dynamics a logarithmic wall function condition is applied to the inner walls of the duct. A velocity of 200 m/s is set at the inlet section and in the outlet section the absolute pressure is set to 101325 Pa. For electrostatic, a constant positive 50 kV DC voltage was applied to the positive electrode and 0V were applied to the other one. For the convection and diffusion model a zero diffusive flux condition is imposed on all boundaries except for the outlet surface where convective flux diffusion is imposed.

Simulation Parameters. The models have been run by COMSOL 3.5a separating a module involving the electrostatics and fluid mechanics from a module that treats the convection and diffusion aspects. The first module has been solved by executing a stationary study, then the results have been used to solve the second module that has been analyzed with a transient study. The Direct (PARDISO) method has been adopted to solve the linear system of the stationary module, and the BDF method for the transient module.

For the stationary solver, a relative tolerance equal to 10^{-6} has been used to control the convergence and the maximum number of iteration has been set to 100. In the transient analysis the relative tolerance has been set to 0.01 and the absolute one equal to 0.001.

Firstly the solution is obtained for the electrostatic and fluidodynamic module with the use of stationary solver. In this way the velocity field and the electric field are calculated and used as source terms to solve the convection and diffusion equations where transient interactive solver are used.

Results and discussions

In this section the results obtained from the simulation model of the magnetohydrodynamics generator are presented. The solution results are presented for 2D configuration model.

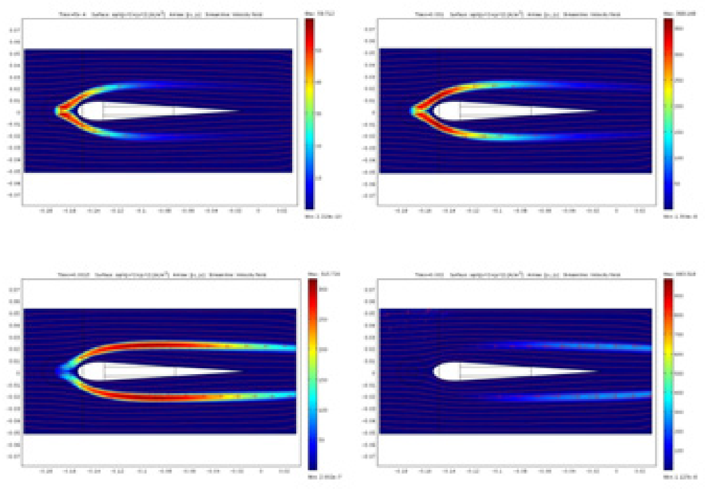

Figure 4: Numerical simulation results

Surface plots of the investigate model shown in Figure 4 display , in the sequential images, the movement of the charge inside of the duct. We can observe the charge flow is spit in two currents one having an excess of positive charge, the other one negative charge. The calculation of the inductive electromotive force has been obtained by considering an equivalent circuit where the primary winding is due to the motion of electric charge and the secondary is from 8000 coils distributed in all other branches of the circuit. The electromagnetic material has been assumed with a relative linear permeability of 40000. By considering the secondary winding left open circuited, has been obtained a tension to the heads of the coil shown in Figure 5. Assuming an adapted load the theoretical maximum power equal to 18kW is obtained.

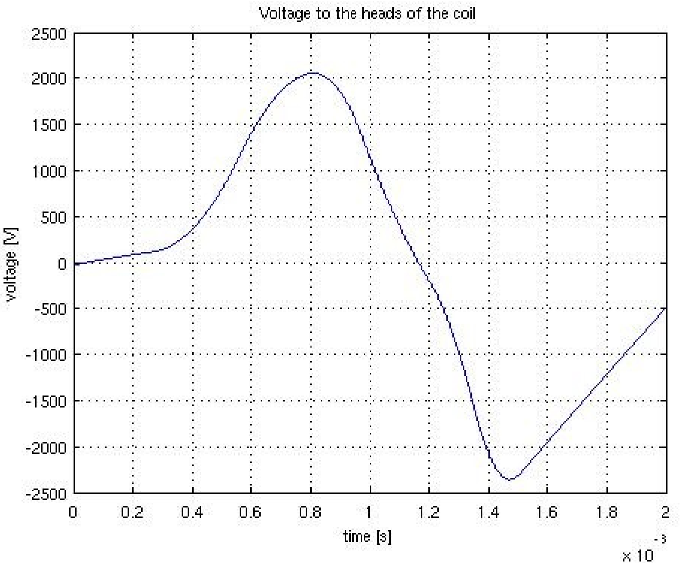

Figure 5: Voltage to the heads of the coil

Conclusions

In this work, a preliminary study has been done to evaluate the physical feasibility of a inductive MHD generator. The proposed device does not need an external magnetic field to work, but it performs the energy conversion by means of the induction principle. This is possible thanks o a pulsed ionization of the fluid current, carried out with an electrode dipped in the fluid current, followed by a charge separation with a stationary electric field. Fluid dynamic, electrostatic and convection and diffusion analysis have been performed by means of Finite Element Method (FEM) where the 2D configuration model has been modeled using COMSOL multiphysics.

References

[1] R.J. Rosa, *"Magnetohydrodynamics Energy Conversion"*, McGraw-Hill, New York (1987).

[2] A. V. Mikheev,N. Kayukawa, N. Okinaka, Y. Kamada, and S. Yatsu, *"High-Temperature Coal-Syngas Plasma Characteristics for Advanced MHD Power Generation"*, IEEE Transactions on Energy Conversion, Vol.21,No. 1, March 2006.

[3] N. Kayukawa, *"Comparison of MHD topping combined power generation systems"*, Energy Conversion & Management 41 (2000) 1953-1974

[4] R.S. Delogu, M. Di Mauro, A. Montisci, *"Inquiry for Technical Feasibility Assessment of a New Design of MHD Generator"* Proc. Of the 13th Biennal IEEE Conference on Electromagnetic Field Computation (CEFC 2006). April 30th – May 3rd .

[5] N. Laser, S. Karpov, I. Krichtafovitch, V. Jayanty, C. Hsu, A. Mamishev, *"Modeling of corona-induced electrohydrodynamics flow with COMSOL multiphysics"*, ESA Annual Meeting on Electrostatics 2008, Paper E1.