FLOW BEHAVIOR IN THE DISK MHD GENERATOR OF THE SUPersonIC
CLOSED LOOP EXPERIMENTAL FACILITY FOR CCMHD POWER
GENERATION

A. Liberati, T. Murakami, Y. Okuno and H. Yamasaki
Tokyo Institute of Technology
4259 Nagatsuta, Midori-ku, Yokohama 226-8502, Japan.
aleslib@es.titech.ac.jp

Abstract. Time dependent r-z two-dimensional numerical simulation has been carried out in order to clarify the
flow behavior inside the disk channel installed in the new closed loop experimental facility at Tokyo Institute of
Technology. The results explain the flow behavior both for high temperature (1800K) and room temperature
(300K) argon gas flow in the facility. In particular, considering boundary conditions obtained by the ongoing
experiments, room temperature argon gas flow behavior has been clarified and the numerical results have been
compared with experimental ones for several values of the mass flow rate; good agreement is shown especially on
the distribution of the stagnation pressure and the mach number at the exit of the generator channel. For high
temperature argon gas flow circulation, the simulation results still evidence the development of thick boundary
layers and the presence of pseudo-shock waves in the core flow in the disk channel. Furthermore, considering the
future MHD application of the generator in the facility, MHD equations have been solved and a first series of
results regarding the performance of the generator are presented also.

Keywords: Close Cycle MHD experimental facility, MHD generator, Disk channel, Large Eddy Simulation.

1. Introduction

A new closed loop experimental facility has been constructed at Tokyo Institute of Technology. The
facility is a closed cycle MHD electrical power generation system without any conventional gas or
steam turbines. The main components of the facility are an oil injection screw type compressor, a
recuperator, a high temperature gas heater with electric heaters, a seed injection system, a liquid
helium free superconducting magnet, a disk shaped MHD generator, an argon cooling duct, a shell and
tube type cooler, a gas purification system. The experimental facility has three important objectives:
demonstration of high temperature gas circulation with heat recovery, long time power generation with a disk
MHD generator, test of durability of each component [1]. Analyses provided key information that led to the
start of the loop design and construction project [2].

Until now some experiments have been already performed in order to test each component of the
facility [3]. Also several kinds of numerical studies have been carried out in order to suggest the start up
operation [4]. This paper is focused on the region of the MHD channel. Figure 1 shows the cross sectional
view of this region. In the channel, the fluid flows downward, axially, in the inlet duct, flows outward,
radially, in the MHD generator channel, and flows downward, axially again, in the cylindrical
downstream channel. The maximum diameter of the region is 380mm. The inlet duct has a 80mm inner
diameter and consists of three ceramics layers and water-cooled stainless steel vessel. The electrodes of
anodes and cathodes are put on the two disk walls and between them there is an insulator wall made by
ceramic. Coaxial inner and outer water-cooled vessels with 292mm and 332mm diameter, respectively, form
the cylindrical downstream channel. The main geometry of the disk channel, however, can be changed varying the height of the throat at the inlet of the divergent channel. In this sense, following the
early experimental results [3] and previous numerical investigations [5], two possible configurations of the
channel have been considered in our work, depending on the temperature of the argon gas circulation in the
facility.

For both the above configuration, a time dependent r-z two-dimensional simulation has been carried out
for the first time in order to investigate the flow behavior in the channel. The results explain the flow
behavior both for high temperature (1800K) and room temperature (300K) argon gas flow. Considering also the results of the early room temperature gas flow experiment, the numerical results have been compared with the experimental ones. The comparison is important to better explain and understand the experimental results, but also to understand if the used numerical model of the simulation gives correct or incorrect results. In these sense, a good agreement let us to extend the same numerical model also to the high temperature flow circulation case. Finally, considering the MHD equations with two-temperature model, in the paper, the latter equations have been solved in order to evaluate for the first time the numerical performance of the generator in the facility. This investigation is particularly important because it could suggest possible improvements of the generator before that the final experiment will be carried out.

2. Numerical Procedure and Conditions

2.1 Basic Navier Stokes equations and LES

In order to investigate the flow behavior in the channel, a large eddy simulation technique has been used to describe the turbulence model. In the Large Eddy Simulation of compressible turbulent flow, any field in the flow domain, are decomposed into the filtered part, directly resolved at the Grid Scales (GS), and the sub-grid scales part that accounts for the Sub Grid Scales (SGS) terms which are modeled. In this simulation an implicit top alt filter has been used. To avoid dealing with double correlation terms including density fluctuation, the Favre-Filter is also introduced as follows:

\[ \widetilde{f} = \frac{\tilde{f}}{\rho} \]  

(1)

Applying the filter to the Navier Stokes equations, we obtain the equations in filtered form:

Continuity Equation

\[ \frac{\partial \tilde{\rho}}{\partial t} + \frac{\partial (\tilde{\rho} \tilde{u}_i)}{\partial x_i} = 0 \]  

(2)

Momentum Equation

\[ \frac{\partial (\tilde{\rho} \tilde{u}_i)}{\partial t} + \frac{\partial (\tilde{\rho} \tilde{u}_j \tilde{u}_i + \tilde{p} \delta_{ij} - \tilde{\sigma}_{ij})}{\partial x_j} = -\frac{\partial \tau_{ij}}{\partial x_j} \]  

(3)

Energy Equation

\[ \frac{\partial (\tilde{Q} + \tilde{p} \tilde{u}_i)}{\partial t} + \frac{\partial (\tilde{Q} \tilde{u}_j - \tilde{\sigma}_{ij} \tilde{u}_j - \tilde{q}_j)}{\partial x_j} = -\frac{\partial (Q_j + \sqrt{\rho} D_j)}{\partial x_j} \]  

(4)

Where \( \tilde{\sigma}_{ij} \) is the filtered molecular stress tensor and, \( \tilde{q}_j \) the filtered heat flux. For the static pressure, the filtered equation of state is defined also. In order to evaluate the molecular viscosity and the heat conductivity, the Sutherland’s formula has been used. The constant Prandtl number is set to 0.66.

2.2 SGS Terms and Basic Assumptions

The terms on the right hand side of the filtered equations are the Sub Grid Scales terms; those have effect on the resolved scales. The SGS viscous diffusion is not included in this calculation because of negligible compared to the others terms. In the SGS viscous term, the Leonard and Cross terms are not considered. Even if the last assumption is true only in weakly compressible turbulent flow, here it is accepted because of the simplicity of the model and the not so high fluctuation of the density, inside the divergent part of the channel [6].

The SGS models are modeled using the conventional Smagorinsky model expanded to compressible flow [7]. In particular, let consider the SGS eddy viscosity model,

\[ \tau_{ij} = -\mu_t \left( \tilde{u}_{ij} + \tilde{u}_{ji} - \frac{2}{3} \delta_{ij} \tilde{H}_{kk} \right) \]  

(5)

With \( \mu_t \) the turbulent eddy viscosity coefficient as follows:

\[ \mu_t = \chi C_s A \left[ 2 \tilde{S} \tilde{S} - \frac{2}{3} (\nabla \cdot \tilde{u})^2 + C_m \left( \frac{f_{max}}{A} \sqrt{|\tilde{u}|} \sqrt{(\nabla \cdot \tilde{u})^2} \right)^2 \right] \]  

(6)

where \( C_s \) is the so-called Smagorinsky Coefficient,
which is set to 0.1 in this simulation [8]. $\Delta$ denotes the filter width. In addition a van Driest’s wall-damping function was used since the effect of the eddy viscosity is damped toward the wall. $S_{ij}$ is the rate of the strain tensor of the resolvable field. In the expression (6), the term $f_{pd}$ depends by the pressure distribution, detecting the spot where the pressure field is disturbed:

$$f_{pd} = \frac{1}{p} \nabla \cdot \left( \nabla \phi \right) \left( S_{ij} + \frac{\partial}{\partial x_j} \right)$$

(7)

The SGS heat flux and SGS turbulent diffusion were modeled as follows, on the analogy of the form of the molecular viscosity terms.

$$Q_j = \frac{C_p \mu_j}{Pr_j} \frac{\partial T_j}{\partial x_j}$$

(8)

$$D_j = -\tau_q \frac{\partial T_j}{\partial x_j}$$

(9)

Where Pr is the turbulent Prandtl number that is set 0.9 in this calculation. It is known that the additional constraints imposed on two-dimensional flow change the nature of the turbulence. 2D simulation is able to simulating the “Large Eddies” of the flow and using LES in our study is correct in this sense. In fact LES is still able to simulate the gross properties of the flow, including total energy generation, dissipation and mean flow velocities by judicious choice of viscosity and diffusion coefficients [9]

### 2.3 MHD equations and numerical method

In order to investigate the MHD flow and the future performance of the Disk MHD generator, other equations are needed. In particular MHD equations with two-temperature model [10], Maxwell equations, together with the equation of state, are used. These equations are shown below.

#### System for charged particles:

A non-equilibrium MHD plasma consists of noble gas atoms, noble gas ions, seed atoms, seed ions, and electrons. The governing equations for charged particles are as follows:

- **Ionization equation**

  $$\frac{\partial n^+_i}{\partial t} + \nabla \cdot (n^+_i \vec{u}^i) = n^+_i \nabla \cdot \vec{u} = k_i n^+_i n_i - k_n n_i n^+_i$$

  (10)

- **Conservation of electron energy**

  $$\frac{\partial n^+_i}{\partial t} = 3n_i \frac{V}{m} \alpha \sum_i n_i \left( \frac{3}{2} k T_i + \epsilon_i \right)$$

  (13)

In the equation (10), $k_i$ and $k_n$ are the ionization rate coefficient of the $i$th particle and the recombination rate coefficient, respectively [11,12].

#### System for Heavy particles

This equations are the Navier Stokes equations (2), (3), and (4), used to describe the flow dynamical behavior in the channel without MHD generation. However, the Lorentz force,

$$\vec{F}_l = \vec{j} \times \vec{B}$$

(14)

is added to the right-hand side of the momentum equation (3), and the joule heating term,

$$\frac{\partial j^i}{\partial t}$$

(15)

to the right-hand side of the energy equation (4).

#### Maxwell Equations

The MHD approximations of charge neutrality and low magnetic Reynolds number are assumed. Then, Maxwell equations are reduced as follows:

$$\nabla \times \vec{E} = 0$$

(16)

$$\nabla \cdot \vec{j} = 0$$

(17)

The ionization equation (10), the continuity equation (2), the momentum equation (3) and the energy equation (4), all of the hyperbolic type, are transformed to a boundary-fitted coordinate, and solved by the CIP method applied to curvilinear coordinate [13] with time step of $dt = 5.0$ns in order to satisfy CFL condition. The non-linear electron energy equation (13) is solved by the bisection method. The substitution of (11) into (17) reduces to the following
elliptic equation

\[
\frac{\partial}{\partial r} \left( \frac{\alpha \phi}{1 + \beta} \right) - \frac{\beta \phi}{r \partial \theta} - \frac{\beta B - \mu B}{1 + \beta} + \frac{\partial}{\partial \theta} \left( \frac{\alpha \phi}{1 + \beta} \right) + \frac{1}{r} \frac{\partial}{\partial r} \left( \frac{\alpha \phi}{1 + \beta} \right) + \frac{\partial}{\partial z} \left( \frac{\alpha \phi}{1 + \beta} \right) = 0
\]  

(18)

The potential function \( \phi \) is defined from (16) as

\[
E_r = \frac{\partial \phi}{\partial r} \quad E_\theta = \frac{\partial \phi}{r \partial \theta} \quad E_z = \frac{\partial \phi}{\partial z}
\]  

(19)

The elliptic equation (18) is discretized with a finite-difference method, and is solved by the Bi-CGSTAB method [14].

2.4 Numerical region and boundary conditions

Figure 2 shows the numerical region used in the calculation. The inner duct has a radius of 40mm. The inner and outer radii of the downstream duct are 146mm and 166mm respectively. The height of throat, \( h_t \), and the height of the exit of the divergent channel \( H_e \) can be changed to different values depending on the temperature of the gas flow in the channel. The radius at the throat and the radius at the disk exit, are 55mm and 115mm respectively. In the case of the room temperature gas circulation in the facility, the disk channel will be called as A in this work, with \( h_t \) set to the value of 2.0mm, and \( H_e \) to 9.0mm (A.R.=9.4). This is the same geometry of the disk channel used in the early experiment of the facility [3]. In the other case of high temperature gas flow circulation, we will call the channel as B, with \( h_t \) set to the value of 3.3mm, and \( H_e \) to 10.3mm (A.R.=6.5). This geometry has been chosen from the results of a previous two-dimensional numerical simulation [5]. The MHD power generation, however, has been simulated only for the channel B.

The boundary conditions used in the simulation are shown in Table 1, Table 2 and Table 3 for the disk channel A and the disk channel B without and with MHD, respectively. Each condition will be discussed in the next paragraphs. For each boundary condition, at the inlet of the channel, fixed value of stagnation pressure and temperature are set. From these, we have obtained the radial and axial velocity, with static pressure and temperature at the inlet by Riemann invariant. At the outlet, the fixed value of the static pressure is used to solve the characteristics equation. Adiabatic wall temperature is set also.

**Table 1: Boundary conditions for channel A: room temperature gas flow. No MHD.**

<table>
<thead>
<tr>
<th>Disk Channel A (h_t=2.0mm, H_e=9.0mm) M=400Nm3/h</th>
<th>Inlet Conditions</th>
<th>T_0=300K – P_0=0.106Mpa</th>
</tr>
</thead>
<tbody>
<tr>
<td>Outlet Conditions</td>
<td>P_outlet= 0.069Mpa</td>
<td></td>
</tr>
<tr>
<td>( P_0/P_{outlet} )</td>
<td>1.54</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Disk Channel A (h_t=2.0mm, H_e=9.0mm) M=450Nm3/h</th>
<th>Inlet Conditions</th>
<th>T_0=300K – P_0=0.118Mpa</th>
</tr>
</thead>
<tbody>
<tr>
<td>Outlet Conditions</td>
<td>P_outlet= 0.052Mpa</td>
<td></td>
</tr>
<tr>
<td>( P_0/P_{outlet} )</td>
<td>2.27</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Disk Channel A (h_t=2.0mm, H_e=9.0mm) M=480Nm3/h</th>
<th>Inlet Conditions</th>
<th>T_0=300K – P_0=0.123Mpa</th>
</tr>
</thead>
<tbody>
<tr>
<td>Outlet Conditions</td>
<td>P_outlet= 0.043Mpa</td>
<td></td>
</tr>
<tr>
<td>( P_0/P_{outlet} )</td>
<td>2.86</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Disk Channel A (h_t=2.0mm, H_e=9.0mm) M=500Nm3/h</th>
<th>Inlet Conditions</th>
<th>T_0=1800K – P_0=0.40MPa</th>
</tr>
</thead>
<tbody>
<tr>
<td>Outlet Conditions</td>
<td>P_{outlet}= 0.148Mpa</td>
<td></td>
</tr>
<tr>
<td>( P_0/P_{outlet} )</td>
<td>3.61</td>
<td></td>
</tr>
</tbody>
</table>

**Table 2: Boundary conditions channel B: high temperature gas flow circulation. No MHD.**

<table>
<thead>
<tr>
<th>Disk Channel B (h_t=3.3mm, H_e=10.3mm) M=540 Nm3/h</th>
<th>Inlet Conditions</th>
<th>T_0=1800K – P_0=0.40MPa</th>
</tr>
</thead>
<tbody>
<tr>
<td>Outlet Conditions</td>
<td>P_{outlet}= 0.125 – 0.148 – 0.155Mpa</td>
<td></td>
</tr>
<tr>
<td>( P_0/P_{outlet} )</td>
<td>3.2 – 2.70 – 2.58</td>
<td></td>
</tr>
</tbody>
</table>

**Table 3: Boundary conditions for channel B with MHD power generation.**

<table>
<thead>
<tr>
<th>Disk Channel B (h_t=3.3mm, H_e=10.3mm) M=540 Nm3/h</th>
<th>Inlet Conditions</th>
<th>T_0=1800K – P_0=0.40MPa</th>
</tr>
</thead>
<tbody>
<tr>
<td>Outlet Conditions</td>
<td>P_{outlet}= 0.148Mpa</td>
<td></td>
</tr>
<tr>
<td>Seed (CS) fraction</td>
<td>4 x 10^-4</td>
<td></td>
</tr>
<tr>
<td>Load Resistance</td>
<td>0.2-0.4-0.8-1.0-1.2 Ω</td>
<td></td>
</tr>
<tr>
<td>Mag. Flux density</td>
<td>4.09 – 4.4 T</td>
<td></td>
</tr>
<tr>
<td>Inlet Electron Te</td>
<td>5000K</td>
<td></td>
</tr>
<tr>
<td>( P_0/P_{outlet} )</td>
<td>2.70</td>
<td></td>
</tr>
</tbody>
</table>
3. Flow behavior for room temperature gas circulation

Table 1 shows the different boundary conditions that we have used in the calculation in order to clarify the flow behavior in the channel for 300K-gas flow. In particular, using the results given directly by the early experiment [3], four different conditions have been considered, respectively with mass flow rates of 400Nm$^3$/h, 450Nm$^3$/h, 480Nm$^3$/h and 500Nm$^3$/h. In the experiment, the stagnation pressure and the Mach number has been measured by a pitot tube, put at the exit of the divergent channel. The position of the Pitot tube is well explained by Fig. 3. Using a micrometer, during the experiment, it has been possible to measure the stagnation pressure, and then the Mach number, for several points of the height of the exit channel’s area. This height is called, in our study, $He$ and it is shown well in Fig. 2. These measurements let us to compare directly the numerical results with the experimental ones, for each studied boundary condition.

Figure 4 shows the Mach number distribution in the channel when the mass flow rate is 400Nm$^3$/h. The flow chokes at the throat and becomes supersonic. However, just downstream at the throat, a shock wave appears in the channel and the flow becomes subsonic with loss of total pressure. The presence of this shock is explained by the low value of the pressure ratio of the inlet stagnation pressure to the outlet static pressure, as shown in Table 1. At the exit of the divergent channel, on the region where the pitot tube is located in the experiment, the subsonic flow let the radial velocity to be small, then the inertial forces are weak, with no deflection of the stagnation pressure. The situation is confirmed by the experimental result of Fig.5 from which is possible to state that the stagnation pressure is mainly constant on the height at the disk exit. From the same figure, it is also possible to confirm that good agreement is present between numerical and experimental results. The same agreement is found on the Mach number distribution of Fig. 6 that, furthermore, confirms the subsonic flow in the channel.

Figure 7 shows the Mach number distribution as the mass flow rate reaches the value of 450Nm$^3$/h. The higher pressure ratio between the inlet and the outlet of the channel lets the flow to have higher Mach number after choking at the throat. The viscous forces and the adverse gradient pressure are responsible for the development of the boundary layer in the upper part of the disk region, downstream to the throat.
However, a weak shock appears again in the channel because of the high value of the outlet static pressure and the flow from supersonic becomes subsonic with loss of total pressure. Furthermore, from Fig. 7, it’s possible to understand that there is a propagation of wavelets in the flow with continuous change of direction of the main flow. At the exit of the channel, the inertial forces induced by the 90°-bend diffuser become important, letting the subsonic flow to be attached in the upper side of the disk wall. This is clarified from Fig. 8 and Fig. 9 that show the deflection of the stagnation pressure and the Mach number distribution at the exit of the disk channel. Also in this situation a good agreement is found between numerical and experimental data.

Raising again the mass flow rate until 480Nm$^3$/h, determines a higher value of the Mach number in the divergent channel as shown in Fig. 10. From this distribution we can observe that the mechanism of pressure loss in the channel remains almost the same, compared to the previous case of Fig. 7. In fact, there is again development of boundary layer with weak shocks in the main flow. Furthermore, at the exit of the disk, the fluid is influenced by the 90° bend diffuser and it is attached in the upper part of the channel as shown in Fig. 10. Experimental results, obtained from the pitot tube, show again a deflection of the stagnation pressure at the exit of the disk. The latter results agree well with the experimental ones, as shown in Fig. 11. Finally, Fig. 12 shows the Mach number distribution at the exit of the channel: good agreement between numerical and experimental data is found also.

At the end, Figure 13 shows the Mach number distribution in the channel when the mass flow rate is 500Nm$^3$/h. The higher value of the pressure ratio
between the inlet and the outlet of the disk, let the flow to be supersonic. Thick boundary layer, however, is developed in the channel. At the exit of the divergent part of the channel, the flow becomes again subsonic because of a series of weak shocks, which appear in the main flow. The adverse gradient pressure induces these shocks. Fig. 14 and Fig. 15, show the distribution of the stagnation and Mach number at the disk exit. Also in this last investigated situation, a good agreement is obtained between numerical and experimental results. It should be interesting to underline that, for room temperature gas circulation, the total pressure loss mechanism is generated by development of boundary layer and presence of weak shock waves, referred as pseudo shock waves [15]. No total pressure loss is evidenced in the 90° bend diffuser.

4. Flow behavior for high temperature gas circulation

Table 2 shows the boundary conditions used in the simulation to study the flow behavior for high temperature gas circulation. The inlet boundaries have been selected from results of a quasi-one dimensional numerical simulation [4]. For the outlet static pressure, three possible values have been selected in order to give a more complete description of the flow in the channel and understand if changing the outlet pressure influences the flow in the channel. Fig. 16, Fig. 17 and Fig. 18 show the Mach number distribution of the flow when the outlet pressure is set to 0.125MPa, 0.148MPa and 0.155MPa respectively. Whatever the value of outlet pressure, the flow chokes upstream to the throat because of thin boundary layers that changes the effective area of the throat. Figure 19 shows the static pressure distribution in the divergent part of the disk channel for $P_{\text{outlet}}=0.125\text{MPa}$ (Fig.19A), $P_{\text{outlet}}=0.148\text{MPa}$ (Fig.19B), $P_{\text{outlet}}=0.155\text{MPa}$ (Fig.19C). When $P_{\text{outlet}}=0.125\text{MPa}$, Figure 16 shows that, after the throat, the fluid is attached on the upper side of the divergent channel with the development of a thick boundary layer. A new detachment of the flow occurs near the disk exit with a new boundary layer developed in the upper part of the channel. In the 90° bend diffuser, an eddy vortex is also evidenced. The reason of this flow pattern can be regarded on the high value of the area ratio (A.R.= 6.5) and the relatively low value of the outlet pressure that influences the flow only at the exit of the divergent channel. As a consequence, there is the detachment of the flow from the upper side of the channel. On the other hand, at the throat, after choking the flow continues to expand and the inertial forces let it to be attached on the upper side of the channel. The mechanism is also evidenced by the static pressure distribution of Fig.19A. A similar Mach number distribution is maintained until the outlet pressure reaches the value of 0.145MPa. However, rising the outlet pressure, the upper
detachment point of the flow moves toward the throat. When $P_{\text{outlet}} = 0.148\text{MPa}$ the flow becomes completely attached on the lower part of the divergent channel. This situation is depicted in Fig.17: the flow chokes upstream to the throat and then a thick boundary layer is developed throughout the divergent channel. The flow is supersonic and the value of the Mach number remains constant. In the $90^\circ$ bend diffuser, a shock wave and an eddy vortex appear and the flow becomes subsonic. The distribution of the pressure, generated by the value of the outlet pressure, plays against the inertial forces, letting the fluid to be attached on the lower part of the divergent channel. Figure 19B evidences this mechanism: the step of the pressure near the throat is obtained with the development of the boundary layer. Rising the back pressure again, weak perturbations such as pseudo shock waves [16] appears into the core flow. When $P_{\text{outlet}} = 0.155\text{MPa}$, as it is shown in Fig. 18, the flow chokes upstream of the throat and then it is attached again on the upper side of the channel and a detachment is present near the exit of the same divergent channel. In this case, the development of the upstream boundary layer is generated by the high value of the outlet pressure. The situation appears similar to that one presented in Fig.16. However the main difference between the Mach distribution of Fig.16 and Fig.18 can be underlined as follows: when $P_{\text{outlet}} = 0.125\text{MPa}$ (relatively small) the inertial forces, influenced by the $90^\circ$ bend diffuser, are responsible for the attachment on the upper side of the channel near the throat. This mechanism is almost the same of that one seen in the low temperature gas flow circulation in the previous paragraph. When $P_{\text{outlet}} = 0.155\text{MPa}$ (relatively high), the high adverse gradient of the pressure, then the pressure forces, are responsible for the attachment on the upper part of the channel. In particular, the adverse gradient of the
Pressure highlights the compressibility effects in the core flow, as it is shown in Fig.19C. Finally, let us consider the eddy vortex and the shock in the 90° bend diffuser. For the situation depicted in Fig. 17, a shock wave is present in the 90° bend diffuser with the flow that becomes subsonic. This means that there aren’t important perturbations inside the divergent channel. In fact the pressure rises in the diffuser. The situation is different in the cases of Fig.16 and Fig.18 where perturbations - not only the boundary layers – such as pseudo shock waves are present in the core flow inside the divergent channel. These conclusions are proved by the value that \( f_{pd} \) reaches in the 90° bend diffuser that is 0.1 for outlet pressure 0.148MPa, 0.06 for outlet pressure 0.125MPa, and 0.03 for outlet pressure 0.155MPa. The presence of pseudo shock waves in the core flow and of boundary layers are important for the future MHD application of the channel [16,17]. For this reason that we have extended our research at the MHD flow in order to obtain a first indication on how the above disk channel works as MHD generator. These results are presented in the next paragraph.

5. MHD flow and performance.

Until now our investigation has been concentrated only on the study of the flow behavior in the channel, with no MHD. In this paragraph, some considerations of the possible MHD flow are presented. Figure 20 show the position of the electrodes in the simulation; the anode is located at \( r = 55 \text{mm} \sim r = 64 \text{mm} \) (9mm width), the cathode starts from the exit of the divergent channel, \( r = 115 \text{mm} \), and it is extended for all the 90° bend diffuser (33mm width). The position of the electrodes is the same as in the experimental facility. However, the width of the anode in the facility is greater, but this doesn’t change our investigation. At this state of the research, we are interested principally on which happens in the divergent channel. Table 3 shows the working conditions used in the simulation. The working gas is argon seeded cesium. The seed fraction is \( 4 \times 10^{-4} \) and it has been selected from previous one-dimensional numerical simulation [18]. The magnetic flux density varies from 4.09T at the downstream edge of the anode to 4.4 T at the upstream edge of the cathode; these are the real values that we can obtain in the experimental facility. Using the two-temperature model, the properties of the heavy particles are calculated from the inlet to the outlet of the whole numerical region as in Fig.2. The electrical values are calculated from the upstream edge of the anode to the downstream edge of the cathode of Fig.20. In the anode region, the electron temperature is assumed to be 5000K, to obtain fully ionization of the seed, and the electron number density is given by the Saha equation [19]. It is clear that in the anode region, the electron temperature is normally lower than 5000K, because of the loading parameter zero. This can induce low electrical conductivity at the exit of the anode and, then ionization instability that, in this first investigation, is not considered. Another important point of the simulation, are the initial conditions for the gas properties [20]. In our work, we have used the initial conditions given by the previous numerical results of flow behavior with no MHD. This choice represents as much as possible the real condition that will be present in the closed loop experimental facility, when MHD power generation experiment will start. Experiment and calculation [21] have shown that long time (few days) needs to reach the suitable conditions of the argon gas for MHD power generation. Then, when MHD starts, the flow in the channel has already its own fluid dynamics. Until now most of the simulation have always set the initial conditions as isentropic. This is correct in most cases, especially for investigation of the simple divergent channel without diffuser, but in our situation, the use of isentropic initial condition is far from the real condition of the flow in the channel.

Several values of the load resistance, \( R_L \), are...
considered: from 0.2 $\Omega$ to 1.2 $\Omega$.

The in-boundary and out-boundary conditions for the main flow has been chosen as in Table 2, with $P_{outlet}$=0.148MPa. This choice has been done because, for the above conditions, with no MHD, the flow is mainly supersonic in the channel, as shown in Fig.17.

The obtained Hall potential distribution in the channel, under different value of the load resistance, is presented in Fig.21. This figure shows that the load resistance influences the MHD interaction and then the plasma behavior in the channel. A similar influence has been found, recently, in experimental results for shock-tube-driven MHD generator [22]. When $R_L=0.2\Omega$, hall potential at the exit of the anode becomes considerably negative. This can be related to the load matching. Thus, the internal resistance in the generator is higher than the fixed external load resistance. A similar condition is maintained when $R_L=0.4\Omega$. However, from $R_L=0.6\Omega$ to $R_L=1.0\Omega$, the Hall potential becomes positive at the inlet of the channel. For these values of $R_L$, an increasing Hall electric field is present in the generator and, for $R_L=0.8\Omega$, the maximum enthalpy extraction ratio (E.E.) of 5.8%, and the maximum isentropic efficiency (I.E.) of 24% (for the highest stagnation pressure at the exit of the generator), are obtained.

The values of E.E. and I.E., against $R_L$, are shown in Fig.22. The thermal input of the generator is 0.45MW, while the maximum power output is 27kW. Considering again Fig.21 and Fig.22, when $R_L=1.2\Omega$, there is a reduction of the Hall output voltage and a reduction of the E.E. and I.E. This can be explained considering Fig. 23 and 24, that show the Mach number distribution in the channel for $R_L=0.8\Omega$ and $R_L=1.2\Omega$ respectively. In the first case, Fig.23, the Lorentz force reduces the Mach number of the flow that becomes subsonic near the inlet of the cathode. Furthermore, a thick boundary layer is still developed in the channel. In the case of $R_L=1.2\Omega$, there is strong MHD interaction at the outlet.
of the anode, with a reduction of the thickness of the boundary layer. The flow becomes subsonic for the presence of a shock wave, as shown in Fig.24. The loss of the stagnation pressure for the presence of the shock is high, then the I.E. falls to 8%, as shown in Fig.22. For the correspondent increasing of the static pressure, the electron temperature decreases to less than 4000K, in the midst of the channel, (as Fig.25 shows) with no good ionization of the seed material. This is responsible for the reduction of the Hall potential, shown in Fig.21.

Furthermore, from Fig. 25 is possible to see that for \( R_L = 0.8 \Omega \), the electron temperature in the main flow is always higher than 4000K with good ionization. However, the electron temperature is not constant in the channel. In fact, in the thick boundary layer and in the main flow, different electron temperatures are present. The situation is depicted in Fig.26.

This difference of electron temperatures is due to the different MHD interactions present in the boundary layer, where the gas radial velocity is low, and in the main flow, where the radial velocity is high. This difference, then, resolves in a different value of the electrical conductivity in the main flow and in the boundary. The gradient of the electrical conductivity, principally, affects the performance of the generator.

At the end, it could be interesting to note that, the maximum value of the E.E., obtained from our simulation and with working conditions of Table 3, is 5.8%. This value is much lower than E.E. > 13%, obtained from another simulation on the same MHD generator and same conditions [23]. In the latter work, however, the numerical region is different, because it considers only the generator channel, i.e. the region from the downstream edge of the anode to the upstream edge of the cathode: it isn’t considered, neither the 90° bend diffuser nor the inlet hot duct.

Also the turbulence model, used in the simulation, is different. The main difference between our study and the other one is in the fluid dynamic property of the flow. In particular, a simulation that is concentrated only in the anode-cathode region, in a MHD channel as that one in Fig.1, cannot give the real results, because the influence of the 90° bend diffuser is important. But, the simulation can be seen as an ideal results. In our investigation, the influence of the diffuser is important: this is responsible of the development of the thick boundary layer that affects the performance of the generator; in fact the MHD interaction in the boundary isn’t as in the main flow. From the above consideration, we can state that, to achieve higher performance of the studied generator, is necessary to modify the channel in order to reduce the development of the boundary.

### 6. Conclusions

Time dependent r-z two-dimensional numerical simulation has been carried out in order to clarify the flow and plasma behavior in the disk MHD generator of the closed loop experimental facility at Tokyo Institute of Technology. Three different conditions have been studied.

For room temperature (300K) argon gas circulation, with no MHD, the total pressure loss, in the disk channel, is generated by development of boundary layer and presence of pseudo-shock waves. No total pressure loss is evidenced in the 90° bend diffuser. Comparing the numerical and experimental results,
for four different value of the mass flow rate of the gas, good agreement is found on the distribution of the stagnation pressure and mach number at the exit of the divergent channel.

For high temperature (1800K) argon gas circulation, with no MHD, the outlet static pressure has an important influence on the flow behavior in the channel. Thick boundary layers are developed in the disk. However, it is shown that is possible to find an optimum value of the outlet pressure for which the flow remains supersonic, with constant Mach number throughout the generator channel. Eddy vortices are evidenced in the 90° bend diffuser.

Finally, for high temperature (1800K) argon gas circulation, with MHD, for $R_\Omega=0.8\Omega$ and $T_i=0.45\text{MW}$, E.E.$=5.8\%$ and I.E.$=24\%$ results from the simulation. Changing the load resistance resolves in a change of the plasma dynamic behavior. In particular for $R_\Omega=0.8\Omega$, a thick boundary layer is developed in the flow, with a difference of the electrical conductivity between the boundary and the main flow. When $R_\Omega=1.2\Omega$, strong MHD interaction is obtained in the channel. The thickness of the boundary is reduced, but a shock wave appears in the generator channel, with loss of the performance for the MHD generator because of no ionization of the seed material in a wide part of the disk MHD generator.

**References**


