J. Plasma Fusion Res. SERIES, Vol. 2 (1999) 482–485

Strong Magnetic Implosion in Spherical Reversed Laser Corona: New Results and New Numerical Schemes for MHD Simulations

SOKOLOV Igor V. and SAKAI Jun-ichi
Laboratory for Plasma Astrophysics, Faculty of Engineering, Toyama University, 3190 Gofuku, Toyama 930-8555 Japan

(Received: 8 December 1998 / Accepted: 26 June 1999)

Abstract

We consider a spherical imploding motion of the plasma in a reversed spherical plasma corona, created by a powerful laser, as a possible way to generate a strong magnetic field. The theoretical analysis and the numerical simulation both show that a Rayleigh-Taylor instability is supressed and magnetic fields as high as 10^4 T may be obtained. The complexity of the new physical problem demands to develop an adequate numerical scheme for its simulation, so that a new approach to the construction of high-resolution TVD numerical scheme for MHD and hydrodynamical simulations via artificial wind concept is also described.

Keywords:

high magnetic field, reversed laser corona, TVD numerical schemes

1. Introduction

Experiments on the high magnetic field generation are proceeded for next to 50 years in different laboratories in the world [1,2]. In order to compress an axial magnetic field a cylindrical imploding liner is used. The high magnetic field generation results from the intensification of the field due to the liner motion perpendicularly to the magnetic field lines. The main limiting effect is small-scale Rayleigh-Taylor instability at the inner boundary of the liner which is induced by the deceleration of this boundary by the counterpressure of the parallel magnetic field.

We consider an alternative configuration in which the magnetic field intensification results from a spherically imploding motion of plasma.

The analysis fulfilled within the model of incompressible perfectly conducting fluid in Section 2 reveals a significant difference from the cylindrical configuration. Although the geometry of spherical compression seemed to be not well matched with the vector geometry of the magnetic field, nevertheless this geometrical difference actually may be useful. Rayleigh-Taylor instability appears to be suppressed, as the results of numerical simulation Section 3 show.

Traditional numerical schemes for solving MHD equations are not too reliable in the presence of very low pressure and strong discontinuities. From the other hand it is not easy to apply modern TVD (total variation diminishing) or Godunov scheme [3] in practical MHD simulations. For MHD equations these scheme become extremely sophisticated (see, for instance [4]) and nevertheless its accuracy is still not evident. That is why we propose and try to explore a new way for the construction of the efficient non-oscillatory shock-capturing schemes for MHD simulations. An artificial

Corresponding author's e-mail: sokolov@ecs.toyama-u.ac.jp

wind concept is introduced. The results of the test calculations for MHD and hydrodynamical shock tube problem are presented in Section 4.

2. Spherical Magnetic Implosion

We use incompressible perfectly conducting fluid model for describing the empty spherical cavity collapse in the initially uniform magnetic field. This model is simplest, however it illustrates the main characteristic features of the phenomenon in hand.

In the absence of the magnetic field the collapse (implosion) of the empty spherical cavity driven by the action of an outer strong pressure P_0 is described by well-known Rayleigh solution for the equations of incompressible fluid motion [5]. The motion is spherically symmetric and the radial velocity of the fluid is directed towards the centre of symmetry. Near the centre, when the radius of the collapsing cavity $R_0(t)$ becomes much less than the initial value of radius $R_0(0)$: $R_0(t) \ll R_0(0)$, the collapse is accompanied by a rapid acceleration of the radial motion and by an abrupt increase in the energy density, so that $\rho(\mathrm{d}R_0/\mathrm{d}t)^2 \gg P_0$, where ρ is a density of the fluid.

Let us to consider now the collapse in a conducting fluid in the presence of a weak magnetic field in the spherically collapsing cavity. As long as the magnetic energy is small: $H^2 \ll \rho(\mathrm{d}R_0/\mathrm{d}t)^2$, one can neglect the unfluence of the magnetic field upon the motion of fluid and find the evolution of the field considering the motion as being given. The resulting distribution of the magnetic field inside and outside of the cavity is as follows:

$$H_{\mathrm{ins}} = H_z = H_0 \frac{R_0^2(0)}{R_0^2(t)}, \; H_{\mathrm{out}} = H_r = H_0 \frac{R_0^2(0)}{r^2} \cos\theta,$$

where r, θ, φ are spherical coordinates, r, $R_0(t) \ll R_0(0)$, the axis of the coordinate system being along the direction of the initial magnetic field H_0, the projection onto this axis is denoted by z index. While the field inside the cavity is spatialy uniform and has only the axial component H_z, the field outside the cavity (that is inside of the fluid) has only radial component H_r.

The radial magnetic field lines are drugged out due to the radial fluid motion. It results in an increase in the field intensity. A physical nature of this effect completely differs from the field intensification in the cylindrical implosion which is due to the fluid motion prependicularly to the magnetic field.

One can also anticipate the saturation of a Rayleigh-Taylor instability, because the fluid deceleration by the magnetic field action is no longer due to the counterpressure of the parallel magnetic field. To the contrary, the field lines in the fluid are perpendicular to the boundary and should stabilize the small-scale perturbations of the boundary.

So the high magnetic field generation in the course of a spherical implosion may be even more effective comparing to traditional schemes. The spherical implosion is always more intense comparing to the cylindrical one, and the Rayleigh-Taylor instability seems not to influence strongly upon the magnetic field compression in spherical geometry.

3. Simulation Results

In order to check the possibility of a practical realization of this effect we perform a numerical simulation in a model of compressible perfect ideally conducting one-fluid gas. The initial-boundary conditions are taken for the configuration of the laser target with reversed laser corona, which may be reasonably proposed for the experiment on strong magnetic field generation.

In simulation the initial gas parameters outside of some appropriate spherical volume are supposed to correspond to the conditions in reversed corona, which is produced by the action of some beams of powerful laser, transmitted into the spherical envelope via some small holes. The presence of residual gas inside of the target is also taken into account. Initially uniform magnetic field is imposed at the target, magnetic pressure being small comparing to the pressure in corona. We also chose a simplest 3D uniform cubic grid in order not to impose the axial symmetry of solution by the use of axisymmetric grid. Only the plane symmetry with respect to the planes $x = 0$, $y = 0$ and $z = 0$ is assumed in simulation and appropriate boundary conditions of symmetry are applyed at these planes.

A typical example of the magnetic field distribution is shown at the Fig. 1 for the time at which the the magnetic field strength becomes maximum. In some variants of simulation the maximum magnetic field pressure amounts to $\sim 3P_0$, where P_0 is the initial plasma pressure in corona. On estimating the value of the $P_0 \sim 2N_0T_0$ for the possible experiment with a powerful laser: $N_0 \sim 10^{29}$ m^{-3}, $T_0 \sim 300$ eV one can find that the values of magnetic field intensity which may be obtained in such a way are as high as 10^4 T.

The form of the volume with a strong field surely is not spherical due to anisotropy in Maxwell stress tensor, it resembles ellipsoid. Nevertheless we do not observe

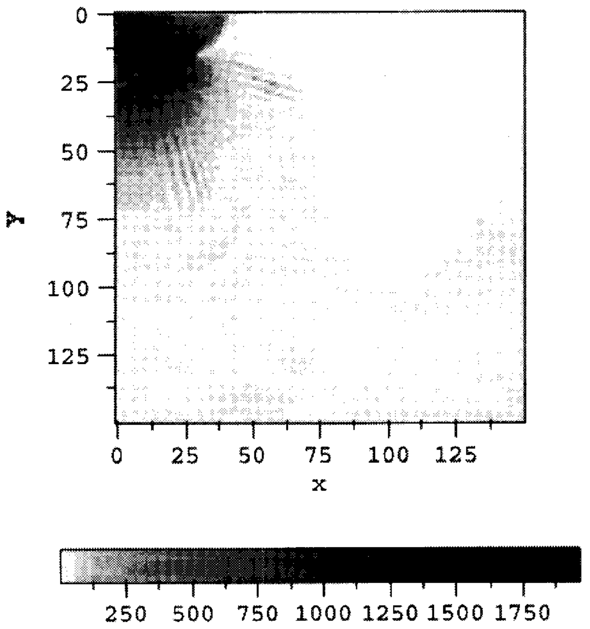

Fig. 1 The distribution of the H^2 in the plane XY. Ideal MHD equations are solved at the cubic grid $150\Delta \times 150\Delta \times 150\Delta$ cells. The size of the cell $\Delta = 1$. Initial radius of the plasma corona $R_0 = 100\Delta$, initial pressure and density for the corona ($r > R_0$) and for the residual gas ($r < R_0$) are $P_c = 1000$, $\rho_c = 1000$ and $P_r = 1$, $\rho_r = 1$ respectively. The initial magnetic field has the only X-component and $H_x = 3(8\pi)^{1/2}$. Time instant $t = 48.0\Delta\,(\rho_r/P_r)^{1/2}$.

any kind of small scale Rayleigh-Taylor instability.

4. New High Resolution MHD Scheme with an Artificial Wind

We hope that the results of the Sections 2–3 are of interest themselves, however they may be also considered as the statement of the new problem for computational physics. Indeed, in order to obtain exact and reliable data for the value of the magnetic field in reversed spherical corona one has to use 3D MHD numerical scheme with high spatial resolution. The reliable estimate of stability of the implosion is possible only in the case when the numerical scheme in stable itself and when it is free of any kind of own ocillations. We do not know the scheme for MHD simulations which would meet well with these requirements.

So we propose here and try to elaborate a new way to construct high-resolution non-oscillatory shock-capturing schemes. The way allows to avoid complexities mentioned in the Introduction and, therefore, to contribute to calculational simplicity and efficiency. At the same time our approach employs TVD limiters leading to oscillation-free solutions. Our key point is an *artificial wind* concept.

We pay attention to the fact that Galilee invariance of hydrodynamic and MHD equations allows to use different steadily moving frames of reference for solving the equations. Moreover, generally speaking the frame of reference may be changed from one time integration step to another and even during one time step, for instance, between the predictor and corrector calculations of a numerical scheme. An additional velocity (*artificial wind*) is added to the velocity of the flow under simulation when the system of coordinates is changed.

The crucial point is that the frame of reference may be chosen in such a way that *all* the flow under simulation is *supersonic* there. In such a situation, the solution of the computational problems typical of high resolution schemes becomes almost trivial.

Let us consider below the simplest case of 1-D gas flows and uniformly spaced (Δx) meshes. Let Δt be the time step value satisfying the CFL condition for a scheme: $\Delta t = \sigma\,\Delta x/C$, where C is the spectral radius of the Jacobian matrix (physically it represents the maximum speed of disturbance propagation); $\sigma < 1$ is the Courant number ensuring stability for the scheme under consideration.

Under the above conventions, a natural way to apply the artificial wind idea to numerical integration is as follows. Let us define the artificial wind velocity as $D = \Delta x/\Delta t$. One can use the frame of reference moving in the positive direction of the x axis at the velocity D during the first time half step $\Delta t/2$ and the frame moving in the opposite direction during the second time half step. The velocity D is added (with a proper sign) to the velocity of the flow under consideration when the transformations into the new system of coordinates are performed. A few important points should be emphasized here. (1) The definition of the velocity D ensures that the flow becomes supersonic with respect to the moving frames of reference and, as it is mentioned above, allows to simplify considerably the scheme in use. (2) It is easy to prove that the CFL condition is hold for each of the half steps either. (3) Since we apply the artificial wind of opposite sign the whole procedure (time step Δt) results in the numerical solution for the original system of coordinates.

Rather simple manipulations with the governing equations omitted here show that one does not need actually to add the velocity D to the flow velocity transforming the conservative variables U to their values in the new frame of reference. It is enough to modify only the expression for fluxes $F: F^D = F - DU$.

The above considerations are of general nature and are not based on a particular scheme. Virtually any scheme can be used within the artificial wind concept. The simplest example is the MacCormack-Upwind scheme with artificial wind (AW) [6]. The promising scheme may be obtained if the time step $\Delta t/2$ in a procedure described above tends to zero, that is the change in the direction of the AW becomes infinitely frequent. For each the boundary between cells the minimum possible amplitudes of the right AW $D^r_{i+1/2}$ and the left AW $D^l_{i+1/2}$ may be chosen $D^l_{i+1/2} = \min(\lambda_i, \lambda_{i+1})$, $D^r_{i+1/2} = \max(\lambda_i, \lambda_{i+1})$, where λ_i are all the characteristic eigenvalues in the adjacent cells. With this choice of the AW velocities TVD a simple conservative scheme is obtained in a form as follows:

$$\frac{dU_i}{dt} = \frac{F_{i-1/2} - F_{i+1/2}}{\Delta x}, \quad F_{i+1/2} = \frac{D^r_{i+1/2}F(U^l_{i+1/2}) - D^l_{i+1/2}F(U^r_{i+1/2}) - D^r_{i+1/2}D^l_{i+1/2}(U^l_{i+1/2} - U^r_{i+1/2})}{D^r_{i+1/2} - D^l_{i+1/2}}$$

where the right and the left extrapolations of the conservative variavles to the boundary $U^r_{i+1/2}$ and $U^l_{i+1/2}$ should be limitted in an usual way. We used a way to find these extrapolated values and predictor for the numerical integration over the time variables which are both described in [7] as well as β-limitter, described in [3].

The results of the test simulation are presented at the Fig. 2. We see that depending of the optimal choice of parameter β in limitter one can obtain either high-quality results for fine grids or reliable high-resolution of the discontinuities at the rough grid, the simplicity and efficiency of the calculations being very high in all the cases.

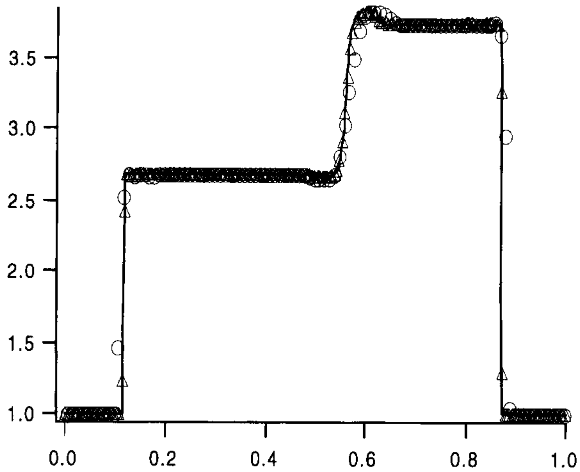

Fig. 2 The results of check MHD test simulations for the numerical scheme with an artificial wind. The test is described in [4] and in papers cited there. Line - 400 cell per the full length (limitter with $\beta = 1.3$, triangles - 200 cells ($\beta = 1.4$), circles - 100 cells ($\beta = 1.7$). The time step correspons to the CFL number $\sigma = 0.8$

5. Conclusions

According to the estimates and the first results of the numerical simulation the spherical implosion of plasma may be considered as the prominent generator of the high magnetic fields. We see also that the results of the numerical simulation for complicated plasma dynamics in 3D implosion should be checked using more modern and reliable scheme for numerical MHD simulation. The way to construct these scheme in a simple and efficient way is found and described.

References

[1] A.D. Sakharov, Sov. Phys. Uspekhi **88**, 725 (1966).

[2] H. Knoepfel, *Pulsed High Magnetic Fields* (North Holland Publ.) 1976.

[3] C. Hirsch, *Numerical Computation of Internal and External Flows.* (John Wiley & Sons) Vol. 2, 1990.

[4] D. Ryu, T.W. Jones and A. Frank, Astrophys. J. **452**, 785 (1995).

[5] L.D. Landau and E.M. Lifshits, *Hydrodynamics* (Wiley, NY) 1994.

[6] I.V. Sokolov, J.I. Sakai and E.V. Timopheev, *Proc. of the 12th National CFD Conf.* (Chuo Univ., Tokyo) 93 (1998).

[7] A.V. Rodionov, Zh. Vychisl, Mat. Fiz. (in Russian) **27**, 1853 (1987).