

MHD Simulations of the Solar Wind Interaction with Non-magnetized Planets

Andreas Ekenbäck*, Mats Holmström*, and Herbert Gunell*

The traditional way of making MHD simulations of the interaction between non-magnetized planets and the solar wind is to define an inner spherical boundary. Often a conducting sphere where the ionospheric currents will be induced. Such a curved boundary presents a problem when using a generic MHD solver on a Cartesian grid. It is not easy to modify the solver at the boundary while preserving the accuracy of the numerical scheme.

For a non-magnetized planet such as Venus or Mars, the obstacle to the solar wind flow is the ionosphere, that is created by the photo-ionization of neutrals. Currents are induced in the ionosphere, and these will deflect the incoming solar wind, creating a bow shock in front of the planet, and an ionopause boundary (at Venus), or an induced magnetospheric boundary (at Mars).

Here we propose an approach where we do not have any inner boundary at all. We only specify sources and sinks of ions close to the planet (in the ionosphere). This allows us to use a Cartesian grid MHD solver.

For an initial study of the concept, instead of building a complete ionospheric model with many species, sources, and loss terms, we use a single fluid MHD model with a source of photoions given by the simplified Chapman production function

$$p(h, \chi) = p_0 e^{1-y-\sec \chi e^{-y}}, \quad y = (h - h_0)/H, \quad h \geq 0, \quad 0 \leq \chi < \pi/2, \quad (1)$$

where h [m] is the height above the planet surface, χ [rad] is the solar zenith angle, p_0 [$\text{m}^{-3}\text{s}^{-1}$] is the maximum production along the sub-solar line, at height h_0 [m], and H [m] is the scale height. Here all model parameters are chosen to be relevant for the Mars-solar wind interaction.

A future extension of the code to the more realistic multi-fluid setting should not present a problem since the MHD solver used can handle that. The code that we have adapted (FLASH from University of Chicago) is an open source application capable of handling general compressible flow problems. The modularity of the code permits users to solve flow problems using only the appropriate modules. Written mainly in Fortran 90, the FLASH code contains different solvers for flow problems along with a version of the PARAMESH library that handles an adaptive computational grid. FLASH is portable, and uses the Message-Passing Interface (MPI) for inter-processor communication. By using this existing code we benefit from the extensive testing and optimization of the code that has been done.

*Swedish Institute of Space Physics (IRF), PO Box 812, SE-981 28 Kiruna, Sweden. andreas.ekenback@irf.se