

DEVELOPMENT OF PROTOTYPE LATTICE-BOLTZMANN CODE FOR COMPUTATIONAL FLUID DYNAMICS (CFD) OF FUSION SYSTEMS

PROJECT SUMMARY

Need for Nuclear Fusion

Nuclear fusion is commonly perceived as being a cheap and unlimited source of energy; certainly its development as a large scale source of power would be a huge step forward in addressing global needs. Currently the bulk of the world's energy consumption is from fossil sources, but the supply of these is finite with many reserves rapidly dwindling, whereas the supply of fusion fuel is virtually unlimited. Another advantage over fossil fuels is that fusion power does not produce carbon dioxide or other pollutants. Unlike nuclear fission, there is little in the way of radioactive waste products and from a political perspective, the process is likely to be less controversial and without the opportunity for arms proliferation. Land usage is much less demanding than renewable energy sources and there is more freedom in where to site power stations – for example, unlike wind and solar power, there are no constraints requiring siting at windy or sunny areas.

Technical Challenges of Fusion

The potential for fusion power has led to a number of major research programs. The ITER project, for example, is an international initiative being undertaken by a consortium of countries, including the US, European Union, Russia, China and Japan. The project has a budget of €5 billion over the next ten years and will involve the construction of a prototype fusion reactor. While the principles behind nuclear fusion are well understood, in practice harnessing this source of energy poses immense engineering hurdles which need to be overcome. The main difficulties arise from the very high temperatures and heat fluxes generated in the reaction chamber, and resilient materials and efficient heat exchangers are required. In addition, many designs for fusion chambers make use of liquid metals in the cooling systems because of their high thermal conductivities. The presence of intense magnetic fields in prototype fusion chambers leads to electric currents in the liquid metals and the resulting electromagnetic forces can considerably affect the flow. This involves the need to model magnetohydrodynamic (MHD) effects.

Role of CFD

Computational fluid dynamics (CFD) tools play an important role in the design of components in prototype fusion chambers. Studies of the cooling systems make use of CFD codes which solve the equations governing fluid flow. To this end, MetaHeuristics has undertaken a project sponsored by the Department of Energy (DOE) under their SBIR program, to develop CFD techniques for liquid metal flows with MHD effects and thus support the ITER project. Almost all existing CFD codes use methods based on the finite difference approach for the solution of the equations. However a newer approach, the lattice Boltzmann method (LBM), pioneered mainly by university researchers, is being used by MetaHeuristics. The LBM has a number of advantages over finite difference solvers. The chief advantage is that it is well suited to parallel processing – CFD problems are computationally demanding and multiprocessor supercomputers are frequently applied to their solution. With existing methods, the proportion of time the processors spend communicating with one other rises as the number of processors increases, thus placing a limit on the number of processors that can be usefully used. LBM codes do not suffer from this drawback and also have a relatively fast execution time, thus enabling higher resolution simulations and a greater range of problems to be run.

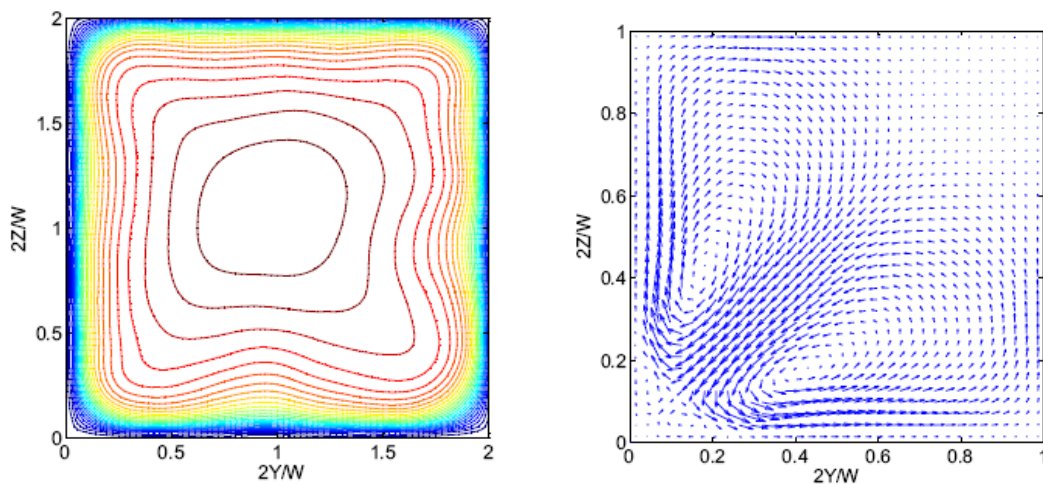
Models and Results

MetaHeuristics has implemented a comprehensive set of state-of-the-art lattice Boltzmann based models in its MetaFlow code. This has involved the development of a number of new models and computational techniques, in particular to deal with MHD effects as very little work had been done in this area.

The core solver for the dynamics of the flow uses a recently developed form of the LBM, the multiple relaxation time (MRT) model. This has proven to perform much better than the single relaxation time (SRT) model used by earlier versions of the code. The MRT overcomes a number of problems, such as stability issues, which may have previously prevented LBM from gaining acceptance in the CFD world. Furthermore, despite its relative complexity, with careful optimisation of the code the MRT only takes about 30% more computer time than

the SRT, and it is expected that MetaFlow will become the first commercial CFD code using the MRT model. Features of the fluid solver are:

- Efficient parallel execution – almost linear scaling found on NSCA supercomputer.
- Fast execution – factor of about three faster than an established finite difference code on a per node per time step basis.
- Large eddy simulation (LES) with Smagorinsky model and wall damping has shown good quantitative agreement with benchmark data, including the challenging case of duct flow, where secondary velocities have been successfully captured.
- Geometry reader imports data from CAD files for complex geometry cases.

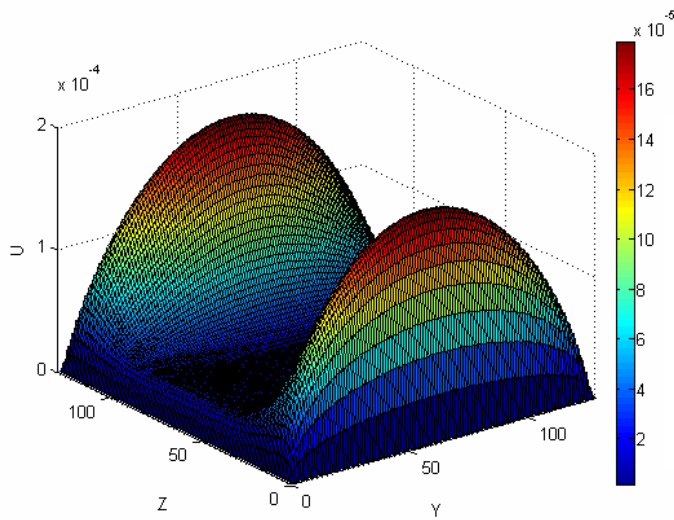


Turbulent flow through a square duct simulated with MetaFlow. On left are isocontours of mean streamwise velocity on duct cross section. Bulging of the contours toward centre is due to the secondary flows moving slow moving fluid away from the wall. On right are mean secondary flows in a quadrant.

The magnetic induction equation is also solved using a lattice Boltzmann approach; this fits in naturally with the scheme used for the equations governing the flow dynamics and permits efficient parallelisation. Relatively little work had previously been done on this method of solving the MHD equations, and a number of new models and computational techniques had to be developed. These included:

- An extrapolation based method for imposing boundary conditions
- A scheme based on a preconditioning approach for allowing low magnetic Prandtl number fluids, such as liquid metals, to be modelled
- Extension of the method to allow stretched grids – particularly important for high Hartmann number flows

The code has been applied to a variety of different flows, including those with insulating walls and ones with conducting walls. In particular, turbulent pipe flow entering a magnetic field has been simulated, with MetaFlow correctly predicting damping of the turbulence.



Velocity profile in duct. Walls along y axis are insulating and those along z axis are conducting. The flow of liquid metals is strongly dependent on MHD effects, and the presence of conducting regions around flow channels can drastically alter the velocity profile and pressure drop.

To complement the above models, a TVD (total variation diminishing) scheme has been developed and implemented to allow heat transfer in, for example, thermal blankets, to be dealt with. The advection/diffusion equation solver can also be applied to passive scalar transport, as may be needed to track tritium levels.

Looking to the future, the techniques developed and the resulting code are likely to find use in a range of applications. As well as heat exchangers for ITER, for which MetaFlow was developed, the code will be applicable to other types of flow, in particular complex turbulent flows common in many engineering applications.