International Conference

Nuclear Energy for New Europe 2004

Portorož • Slovenia • September 6-9

port2004@ijs.si
www.drustvo-js.si/port2004
+386 1 588 5247, fax +386 1 561 2276
PORT2004, Nuclear Society of Slovenia, Jamova 39, SI-1000 Ljubljana, Slovenia

Hydrodynamics around a Spacer of a VVER-440 Fuel Rod Bundle

Gusztáv Mayer, Gábor Házi[1], Péter Kávrán
KFKI Atomic Energy Research Insitute
P.O. Box 49, H-1525 Budapest, Hungary
mayer@sunserv.kfki.hu, gah@sunserv.kfki.hu, kavranp@sunserv.kfki.hu

ABSTRACT

Recently, an intensive research has been started in our institute, focusing on the hydrodynamics of fuel rod bundles. Numerical computations have been planed to be carried out in a three level bottom-up hierarchy, using direct numerical simulation, large eddy simulation and Reynolds averaged Navier-Stokes approach. Here, we give a description of the numerical method applied for direct numerical and large eddy simulation. We present some preliminary results obtained by the simulation of the flow around a spacer of a VVER-440 fuel rod bundle.

1 INTRODUCTION

Advances in computational fluid dynamics (CFD) and the reduction of the cost of high performance numerical computations motivated many researchers to apply numerical methods to study the hydrodynamics of fuel rod bundles [1-7]. Numerical experiments are usually based on the Reynolds averaged Navier-Stokes (RANS) approach, modeling the turbulence with some simple model. The success of such simulations is limited. Recently, Tzanos presented RANS calculations using various clones of k-epsilon models and concluded that those models need to be improved in order to achieve reasonable accuracy in case of rod bundle simulations [1-4].

It is well known fact that simple eddy viscosity models like the k-epsilon performs badly in case of the presence of secondary flow, flow separation and reattachment. Therefore the conclusion given by Tzanos is not surprising and can be drawn *a priori* without any numerical simulation. It might be more surprising that the application of Reynolds stress models provided practically the same results in Tzanos calculations [3].

In this year an intensive research has been started in our institute in order to qualify – and improve if it is possible - the performance of commercial CFD codes for rod bundle simulations. The focus is on VVER-440 rod bundles, i.e. triangular rod arrangement considering the grid spacers, too. The pitch to diameter ratio is roughly 1.3.

A three-level hierarchy of computations has been built up including direct numerical simulation (DNS) in the lowest level as a modeless approach. Due to the computational cost, DNS is limited to study low Reynolds number flows and small piece of the rod bundle

[1] Corresponding author

domain. Practically, DNS is used to study the physics of turbulence in a part of a subchannel of triangular rod bundle and the flow around a fuel spacer. In highest level RANS calculations will be used at nominal flow considering the assembly as a whole including the head, the foot and the separation lattices. Considering accuracy and Reynolds number, the gap between the DNS and RANS calculations is filled up by using large eddy simulations (LES) for subchannel analysis. The results obtained at different levels are compared with each other in order to validate and improve the models applied. The reader not familiar with turbulent flow modeling can look the above strategies up in [8].

Preliminary results have already been obtained in each level but, in this paper we shall focus on the direct and large eddy simulations. The numerical method applied is presented. Discussing some simulation results we outline how the information obtained in DNS can be used to improve the LES results and on the other hand how the LES computations allow reducing the computational cost of DNS. Some conclusions are drawn from the information already in hand and our future work is outlined.

2 NUMERICAL METHOD

For DNS and LES practically the same numerical method has been used in our computations. Differences, like turbulence modelling in case of LES and wall boundary treatment is discussed in the subsequent sections.

2.1 The lattice Boltzmann method

Our approach is based on the lattice Boltzmann method, which can be considered as second-order accurate (both in time and space) Navier-Stokes solver. The theoretical background of this method has been reviewed e.g. in [9] and here we give only the specific information needs to reproduce our simulation results.

The geometrical domain considered (Fig. 1) is covered by a regular mesh using cubes.

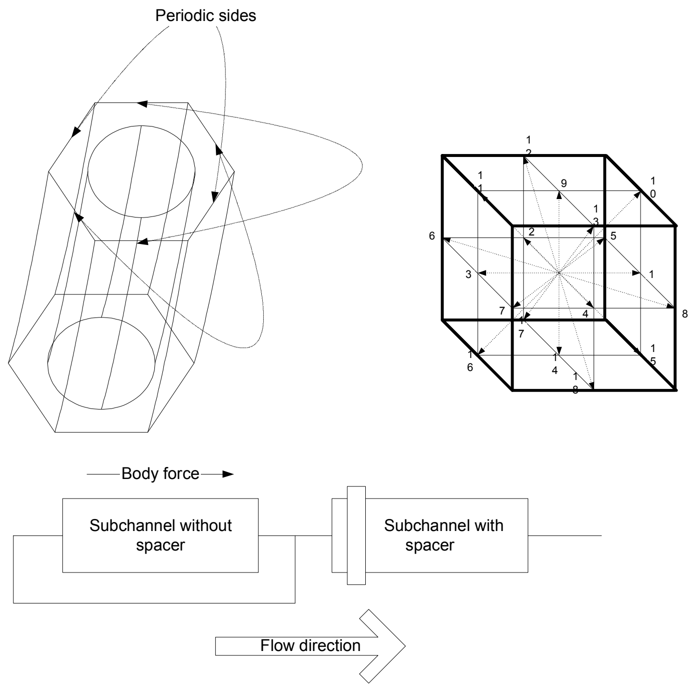

Figure 1: Part of the subchannel of a VVER-440 rod bundle (left). D3Q19 lattice (right). Simulation of the near-spacer regions (bottom).

A D3Q19 model is applied [10], i.e. for each cube nineteen one particle velocity distribution functions are assigned. The evolution of the particle distribution functions is specified by the lattice Boltzmann equation using the Bhatnagar-Gross-Krook collision operator:

$$f_i(r+\delta c_i, t+\delta) - f_i(r,t) = -\frac{1}{\tau}\left[f_i(r,t) - f_i^{eq}(r,t)\right], \tag{1}$$

where $f_i(r,t)$ is i-th the particle distribution function, c_i is the lattice direction, δ is the lattice spacing, τ is the relaxation time and the equilibrium distribution function $f_i^{eq}(r,t)$ is given as follows (repeated indices imply summation):

$$f_i^{eq}(r,t) = w_i \rho \left[1 + \frac{c_{i\alpha} u_\alpha}{c_s^2} + \frac{u_\alpha u_\beta}{2c_s^4}\left(c_{i\alpha} c_{i\beta} - c_s^2 \delta_{\alpha\beta}\right)\right], \tag{2}$$

where w is the lattice weight [10]. The macroscopic quantities can be obtained by taking the suitable moments of the distribution functions:

$$\rho = \sum_i f_i(r,t), \qquad \rho u_\alpha = \sum_i c_{i\alpha} f_i(r,t), \tag{3}$$

where ρ is the density and u_α is the hydrodynamic velocity.

Solving the lattice Boltzmann equation and computing the macroscopic quantities by the above moments, it can be shown that the velocity satisfy the incompressible Navier-Stokes equations in the low-Mach number limit. The ideal gas law makes relation between the pressure and density:

$$p = \rho c_s^2, \tag{4}$$

where the speed of sound is constant $c_s = 1/\sqrt{3}$ for this model. The kinematic viscosity depends on the lattice spacing and the relaxation time as follows:

$$\nu = \delta c_s^2 (\tau - 0.5). \tag{5}$$

What makes the lattice Boltzmann method so attractive is its algorithm simplicity using a streaming and collision procedure as discussed below.

The lattice Boltzmann equation can be solved by the following algorithm:

1. Collision - relax the distribution functions according to their computed equilibrium values (RHS of Eq. (1)).
2. Streaming - move each distribution function with one lattice according to its direction.
3. Calculate the macroscopic quantities by the moments Eq. (3).

Steps 1-3 have to be sequential but in each step the operations are local and therefore can be parallelized easily using e.g. OpenMP pragmas or more elaborated domain decomposition strategies.

2.2 Boundary conditions

Boundary conditions bring some difficulties into the implementation of the lattice Boltzmann algorithm. In our computations we used an interpolation supplemented bounceback boundary condition [11] to achieve no-slip at the walls. Since in its original form this boundary condition is not mass conservative we modified the method introducing a mass correction. Basically, the mass deflection is added to the rest particle distribution function of the fluid node next to the wall node. From physical point of view this contribution can be considered as a part of particles which do not leave the near-wall region in a mean sense. With the above treatment of no-slip walls the mesh becomes boundary fitted, and consistently with the accuracy of the basic method, second order accuracy can be achieved.

In RANS subchannel calculations, symmetry boundary condition is usually applied. It is reasonable, since in high Reynolds number flows the symmetry properties of the Navier-Stokes equation are restored at least in a statistical sense. Since the RANS approach is based on time-averaged equations, the application of symmetry boundary conditions is justified. However, in case of DNS and LES, the symmetry of the geometry can not be explored and other treatment is needed. Accordingly, as it can be seen in Fig. 1 we modelled not only a cell of but the whole subchannel. Periodic boundary conditions were used both in the axial and radial directions. The implementation of periodic boundary conditions is straightforward in lattice Boltzmann framework.

Since the computational demand both in case of LES and DNS highly limit the size of the geometrical domain, therefore periodic boundary conditions can not be applied axially to study the near spacer regions (the distance between the spacers is roughly 25cm). Therefore, we had to focus only a small part of the subchannel around the spacer. In order to supply realistic inlet flow for the near-spacer regions we used a coupling between a spacer-free subchannel calculation and the near-spacer calculation (Fig. 1). The inlet and the outlet of the spacer-free subchannel are periodically coupled to each other and its outlet provides the inlet for the near-spacer domain. Accordingly, inlet velocity boundary condition also has been implemented for such simulations. Constant pressure at the outlet was applied for the near-spacer domain. For the implementation we used the extrapolation method proposed in [11].

In the spacer-free subchannel the flow is driven by a body force, which can be considered as a pressure gradient and introduced as a simple source term into the lattice Boltzmann equation.

2.3 Turbulence modelling

For LES we have to model the unresolved, so-called subgrid scales (in DNS we have to resolve all energy containing scales and the approach is modeless). In our LES computations we used eddy viscosity concept with the static Smagorinsky subgrid-scale model. The implementation of this model in lattice Boltzmann framework is discussed in [12]. Note that using the lattice Boltzmann method the rate of strain can be computed from the second moment of the non-equilibrium distribution functions. That is, we still can work with local operations and do not need to determine the derivatives of the velocity with finite differences as in case of traditional approaches.

2.4 Related issues

After briefly introducing our numerical approach some comments are in order.

Remark 1 *Accuracy and effectivity:* It is generally accepted that for DNS highly accurate numerical methods are needed. Therefore, in the past, spectral methods were applied almost exclusively. However, due to development of other numerical techniques, today it is not difficult to find DNS in the literature, where less accurate but more effective techniques, like finite volume or kinetic approaches are in action [13]. Actually, a limit exists, where the less accurate but more effective numerical methods should be preferred. Our recent comparison between the lattice Boltzmann method and a pseudospectral method for the simulation of two-dimensional decaying turbulence showed only minor advantages of the spectral methods over the lattice Boltzmann method in terms of accuracy (using the same number of grid points). It is also worth emphasizing, that spectral methods can not be used for complex wall bounded flows, so the application of the lattice Boltzmann method seems to be justified. Nevertheless, the development of a spectral element code is currently going on in our institute in order to cross validate our simulation results.

Remark 2 *Treatment of walls:* In high Reynolds number wall bounded turbulent flows the treatment of near-wall regions is crucial. Basically, in DNS we have to well resolve this region. In LES, to keep the computational cost reasonable, one has to use clustered grids or approximate boundary conditions (e.g. based on the law of the wall). Both approaches have their pros and contras. However, it has to keep in mind that the law of the wall is strictly valid for plane walls and it can be applied only in a mean sense. Accordingly, the application of instantaneous law of the wall (and some of its clones) for LES calculations is not accepted as a generally well working technique. There is an intensive research in this direction. In spite of this fact, one gradient we still need to introduce into our framework is a kind of proper treatment of walls in case of LES. Considering this point we can say that the LES and DNS have to go hand in hand just like for simple flows (flow between parallel plates) they have already done.

Remark 3 *Turbulence modelling in LES:* For LES the most popular subgrid-scale model was introduced into our framework. In case of LES the rule of the turbulence model is less relevant than in case of RANS calculation. First, because the mesh has to be fine enough to resolve the largest energy containing eddies well and second, the unresolved scales, hopefully, have universal characteristics if the grid size is inside the energy cascade. Accordingly, in this region a model, which dissipates reasonable amount of energy, has to work well. However, we have to keep in mind, that there are many applications, which proved that the application of the theoretical value of the Smagorinsky constant is often lead to unreasonable results and a tuning (or dynamic adjustment) of this parameter is needed. This is again a point, where DNS and LES results have to be used parallel, since by the means of DNS results we can test a priori the subgrid scale models and new models can be worked out accordingly the results.

3 FLOW IN SUBCHANNELS AROUND FUEL SPACERS

Here we show some simulation results obtained by using the numerical method introduced in the previous section for the simulation of a subchannel with and without fuel spacer. Because of the space limitation of this paper, a detailed analysis of the results, including quantitative discussion, will be presented elsewhere in the near future.

In order to demonstrate the applicability of our method for the simulation of highly turbulent flows, we show the instantaneous pressure and axial velocity field around a fuel

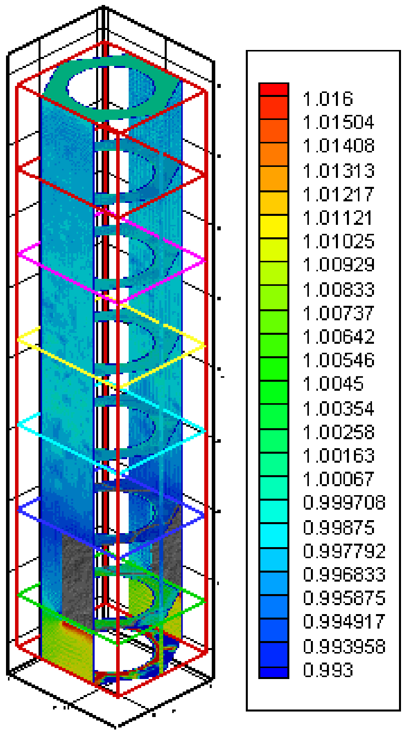

Figure 2**:** Instantaneous pressure field around a fuel spacer. The flow steps into the domain at the bottom. The wall of the spacer is marked by gray color.

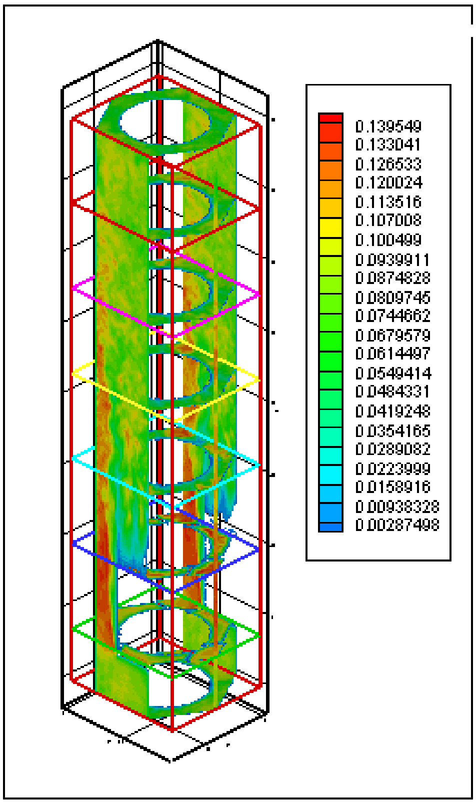

Figure 3**:** Instantaneous axial velocity field around a fuel spacer.

spacer in Fig. 2 and 3. All quantities presented in the figures are dimensionless. Due to the form loss there is a high pressure gradient axially comparing with the amplitude of the background pressure fluctuations, which correspondingly can not be seen clearly. However in Fig. 3 the velocity field shows that the flow is turbulent. Because of some technical problems in visualization the colorbar starts from a positive value and some holes appear near the top of the spacer. In these holes the velocity is negative, showing some recirculation zones. It can be seen in Fig. 3 that the flow is accelerated in the holes of the spacer and directly above the spacer the fluctuation of the velocity much more relevant than at the inlet (which is driven by a subchannel calculation without spacer). With the parameters used for this simulation the flow settles down soon, not too far from the spacer. It is worth emphasizing that only after the analysis of the statistical quantities (e.g. by studying the axial variation of the mean axial velocity) can be drawn any conclusion from these simulations. During the analysis our results have to be confronted with the available experimental results.

To be more informative let us summarize some features, what is expected to be recovered by the numerical simulations:

1. In subchannel calculations the mean axial velocity profile is well known from measurements [15].
2. Secondary flow is observed in some measurements [15] and recently analytical results justified its existence [14] in subchannel cells.
3. Detailed measurements have been presented for Reynolds stresses in [17].
4. Drag, turbulent intensity etc. was measured for VVER-440 fuel rod spacers in [6].
5. Experimental results show that in VVER-440 bundles, the flow does not become fully developed between two spacers [6].

Finally, let us give an example how the LES calculations can help to reduce the computational cost of DNS. It is well known that the channel has to be long enough for periodic plane channel flows, in order to obtain realistic results [18]. Our trials have already shown that the channel length plays a crucial role in the simulation of subchannels, too. Parametric studies can be carried out in LES with a much lower cost than in DNS, therefore LES calculations can clarify, how long the subchannel studied should be.

4 CONCLUSION AND FUTURE WORK

A numerical approach has been presented for the direct and large eddy simulation of fuel rod bundles. Preliminary results show that the technique introduced here can be used to obtain qualitatively correct results. The key features of subchannel flows and flow around a fuel spacer has been listed. Our future work has to focus on the analysis of the statistical quantities in order to demonstrate that the key features can be reproduced.

REFERENCES

[1] C. Tzanos, "Performance of k-epsilon Turbulence Models in the Simulation of LWR Fuel-Bundle Flows", *Trans. Am. Nucl. Soc.*, **84**, 2001, pp. 197-199

[2] C. Tzanos, "Large Eddy Simulation of Flow in LWR Fuel Bundles", *Trans. Am. Nucl. Soc.*, **85**, 2001, pp. 261-262

[3] C. Tzanos, "Performance of the RNG and Two-layer k-epsilon Models int the Simulation of LWR Fuel-Bundle Flows ", *Trans. Am. Nucl. Soc.*, **85**, 2002, pp. 148-150

[4] C. Tzanos, "Computational Fluid Dynamics for the Analysis of Light Water Reactor Flows", *Nucl. Tech.*, **147,** 2004, pp. 181-190

[5] V. Kriventsev, H. Ninokata, "Calculation of Detailed Velocity and Temperature Distributions in a Rod Bundle of Nuclear Reactor", Proc. IX. Int. Top. Meeting Nucl. Reac. Therm. Hydr. NURETH-9, San Francisco, California, October 3-8, 1999

[6] V. Lestinen, P. Gango, "Experimental and Numerical Studies of the Flow Field Characteristics of VVER-440 Fuel Assembly", Proc. IX. Int. Top. Meeting Nucl. Reac. Therm. Hydr. NURETH-9, San Francisco, California, October 3-8, 1999

[7] P. Rautaheimo, E. Salminen, T. Siikonen, J. Hyvarinen, "Turbulent Mixing Between VVER-440 Fuel Bundle Subchannels: A CFD Study", Proc. IX. Int. Top. Meeting Nucl. Reac. Therm. Hydr. NURETH-9, San Francisco, California, October 3-8, 1999

[8] S.B. Pope, Turbulent Flows, Cambridge Univ. Press, Cambridge, 2001

[9] G. Házi, A.R. Imre, G. Mayer, I. Farkas, "Lattice Boltzmann Method for Two-Phase Flow Modeling", *Ann. Nucl. Energy*, **29,** 2002, pp. 1421-1453

[10] Y.H. Qian, D. d'Humiéres, P. Lallemand, "Lattice BGK Models for Navier-Stokes Equation", *Europhys. Lett.* **17,** 1992, pp. 479-484

[11] D. Yu, Viscous Flow Computations with the Lattice Boltzmann Equation Method, Ph.D. Thesis, Univ. Florida, USA, 2002

[12] S. Hou, J. Sterling, S. Chen, G.D. Doolen, "A Lattice Boltzmann Subgrid Model for High Reynolds Number Flows", *Fields Inst. Comm.*, **6**, 1996, pp. 151-165

[13] C. Moulinec, M.J.B.M. Pourquie, B.J. Boersma, T. Buchal, F.T.M. Nieuwstadt, "Direct Numerical Simulation on a Cartesian Mesh of the Flow through a Tube Bundle", *Int. J. Comp. Fluid. Dyn.*, **18,** 2004, pp. 1-14

[14] S. Kim, M.C. Kim, "A Note on the Azimuthal Momentum Transport in Infinitely Arrayed Rod Bundles", *Int. Comm. Heat Mass Tr.*, **31,** 2004, pp. 333-342

[15] A.C. Trupp, R.S. Azad, "The Structure of Turbulent Flow in Triangular Array Rod Bundles", *Nucl. Eng. Des.*, **32,** 1975, pp. 47-84

[16] K.B. Lee, "Analytical Prediction of Subchannel Friction Factor for Infinite Bare Rod Square and Triangular Arrays of Low Pitch to Diameter Ratio in Turbulent Flow", *Nucl. Eng. Des.*, **157,** 1995, pp. 197-203

[17] T. Krauss, L. Meyer, "Experimental Investigation of Turbulent Transport of Momentum and Energy in a Heated Rod Bundle", *Nucl. Eng. Des.*, **180,** 1998, pp. 185-206

[18] J. Jimenez, P. Moin, "The Minimal Flow Unit in Near-Wall Turbulence", *J. Fluid. Mech.*, **225,** 1991, pp. 213-240