ANISOTROPIC POROSITY FORMULATION OF THE COARSE-GRID-CFD (CGCFD)

Mathias Viellieber
Karlsruhe Institute of Technology (KIT)
Institute for Nuclear and Energy Technologies (IKET)
76131 Karlsruhe
Germany
Email: Mathias.Viellieber@kit.edu

Andreas G. Class
Karlsruhe Institute of Technology (KIT)
AREVA Nuclear Professional School (ANPP)
76131 Karlsruhe
Germany
Email: Andreas.Class@kit.edu

ABSTRACT
Computational fluid dynamics (CFD) simulations for large complex geometries such as a complete reactor core of a nuclear power plant requires exceedingly huge computational resources. State of the art computational power and CFD software is restricted to simulations of representative sections of these geometries. The conventional approach to simulate such complex geometries is 1D subchannel analysis employing experimental correlations in the transport models. With the development of the Coarse-Grid-CFD (CGCFD) ([1], [4], [7]), an alternative to the traditional 1D subchannel analysis becomes available which does not need empirical correlations nor specific model constants. The CGCFD approach is based on strongly under-resolved CFD and the inviscid Euler equations. Although the use of the Euler equations and coarse grids does not resolve the subgrid physics like viscous dissipation or turbulence, the subgrid physical information is taken into account by volumetric source terms derived from fully resolved CFD simulations. Non-resolved geometrical information due to the use of very coarse meshes is taken into account by volume porosities and directional surface permeabilities in the finite volume scheme. The volume porosity is defined as the ratio of the control volume that is occupied by the fluid compared to the complete control volume. The surface permeability is defined as the ratio of the individual control surface that is unobstructed to fluid flow compared to the corresponding complete control surface. Due to the use of the volumetric source terms that are derived from fully resolved CFD simulations, distributed resistance used in standart porous media approaches may be omitted and instead be resolved through the volumetric source terms. This is advantageous because the friction factor normally is not very well known in thermal-hydraulic problems and must be derived from comprehensive experiments. Such an anisotropic porosity formulation was originally used in the COMMIX [6] code which was designed to compute complex flow applications in a time when computational resources were limited. The benefits and limitations of our technique are explored by simulating a section of a water rod bundle containing a spacer. General recommendations for the proper application of our technique are presented in this work.

NOMENCLATURE

\( \vec{v} \)  Velocity vector \([m/s]\)
\( \rho \)  Density \([kg/m^3]\)
\( \rho_p \)  Pressure \([Pa]\)
\( k \)  Turbulent kinetic energy \([m^2/s^2]\)
\( \sigma \)  Stress tensor \([N/m^2]\)
\( \mu \)  Dynamic viscosity \([kg/(m/s)]\)
\( \sigma_t \)  Reynolds stress tensor \([N/m^2]\)

*Address all correspondence to this author.


\[ \mu \] Turbulent viscosity \( \frac{[kg]}{[m/s]} \)

\[ \sigma_{ij} \] Kroenecker symbol \([-]\)

\[ n \] Surface normal vector \([-]\)

\[ F_C \] Volumetric force term \([N]\)

\[ \gamma_V \] Volume porosity \([-]\)

\[ \gamma_I \] Surface permeability \([-]\)

\[ R_i \] Distributet resistance \( \frac{[N]}{[m^2]} \)

\[ A_F \] Unobstructed control surface \( [m^2] \)

\[ A_C \] Complete control surface \( [m^2] \)

\[ V_F \] Unobstructed part volume element \( [m^3] \)

\[ V_C \] Complete volume element \( [m^3] \)

INTRODUCTION

One of the most essential parts within a nuclear power plant is the reactor core. Inside the reactor core the nuclear chain reaction takes place and heat is produced that has to be transported by the coolant. Typically the reactor core consists of various fuel assemblies that in turn consists of a multiplicity of fuel rods. For safety reasons, the understanding of the thermohydraulics inside the reactor core is eminent. To improve the knowledge of the thermohydraulics behaviour inside a reactor core, Computational Fluid Dynamics (CFD) simulations are a commonly used tool. The application range of the CFD simulation varies from single subchannels to multiple subchannels. Even whole fuel assemblies are simulated to perform safety relevant CFD analyses. The problem with these safety relevant CFD analyses is that due to computational restraints, only parts of the complete reactor core can be analyzed. This is due to the large numbers of fuel assemblies inside a reactor core, and the numerous length scales that have to be resolved within the reactor core. The resolution must cover the boundary layers of each fuel rod, the spacer or wire wraps up to the complete reactor pressure vessel. For simulations of the complete reactor core either system codes or homogenization was applied. Homogenization often used the porous media equations and the thermohydraulics were averaged over the complete reactor core. System codes used one dimensional conservation equations that were tuned by empirical and geometrical model constants to predict the complex flow situation inside the reactor core.

Currently, several teams like the NRG in the Netherlands are working on new approaches of applying CFD to fuel bundles (compare [7]). The goal of these new approaches is to leave the conventional way by using larger and larger meshes to simulate the complex flow situations inside the reactor core and to try to decrease the number of cells of the computing meshes.

Within this paper we describe the Coarse-Grid-CFD (CGCFD) method, that is developed at the Institute of Nuclear and Energy Technologies (IKET) at the Karlsruhe Institute of Technology (KIT). With the CGCFD we present a new method to simulate the thermohydraulics inside a complete reactor core and give an alternative to the subchannel codes. We propose to combine the ideas of sub channel analysis and CFD and develop the new methodology known as CGCFD which takes advantage of the fast development of commercial CFD software and the efficiency of sub channel analysis. The method uses the inviscid Euler equations and very coarse meshes. Of course, this approach alone can not resolve all the relevant flow phenomena and flow physics. Therefore we tune the Euler equations with a volumetric force term, that is derived from a detailed CFD simulation of a section of the complete geometry that has to be calculated with the CGCFD. The parameterization of these subgrid forces can be realized analogous to the models within the subchannel analysis. The method was first applied to a Generation IV reactor core where the temperature distribution and the pressure drop inside a rod bundle section was analyzed with promising results [1]. In addition to this work, there was an investigation within the framework of accelerator driven sub-critical reactor systems (ADS), where heavy-liquid-metal (HLM) cooled fuel assemblies were considered. At the Karlsruhe Liquid metal Laboratory (KALLA), a series of experiments to quantify both pressure losses and heat transfer in HLM-cooled rod bundles were performed. These experimental investigations were accompanied by detailed CFD investigations of the used rod bundles. For the development of the CGCFD, we used the gained data as a basis for our development of the CGCFD. Within this investigation, the CGCFD was implemented into the open source software OpenFOAM. First results where promising, but within the spacer region of the investigated rod bundle an adjustment factor was used to reproduce the correct pressure drop. The CGCFD should be a general methodology, that is not based on empirical model constants, nor on experimental input. Therefore we consider to combine the CGCFD with an anisotropic porosity formulation, that was originally implemented in the COMMIX [6] code to avoid adjustment factors for specific parts of a geometry. The COMMIX code with the anisotropic porosity formulation was designed to compute complex flow applications in a time when computational resources were limited. It uses a special implementation of an anisotropic porosity formulation that is based on directional surface permeabilities and volume porosities. Within this work, we present the benefits and some limitations of our technique of the CGCFD and the CGCFD in combination with the anisotropic porosity formulation. The new technique is explored by simulating a section of a water rod bundle experiment that was part of the HLM fuel rod investigation and was performed at the KALLA laboratory.

In the first part of the paper, the general idea of the CGCFD is explained. Also a principal methodology and the theory behind the CGCFD is conducted. Then, a short overview is given about the
water rod bundle experiment and the detailed RANS CFD simulations that where performed at the IKET and shows the CGCFD results of this rod bundle without the anisotropic porosity formulation. In the second part of the paper the theory of the CGCFD in combination with the anisotropic porosity formulation is explained and the results of the simulation of the water rod bundle experiment with the new approach are presented.

THE COARSE-GRID-CFD (CGCFD) APPROACH

This section of the paper contains a general overview of the idea of the CGCFD. Especially the motivation of using the CGCFD, the theory and a general methodology for the use of the CGCFD are particularly discussed.

Why use the CGCFD

With state of the art computational resources it is possible to perform a wide range of numerical simulations. Depending on the problem that is being investigated, there are different types of CFD simulations with different types and amount of models that are used to get accurate results of the simulation and of course with different requirements to the computational resources. Figure 1 showing the hierarchy of CFD-simulations that are commonly used to simulate thermohydraulic problems within the nuclear sector. Figure 1 also shows two principal types of thermohydraulic simulations. Simulations using the 3 dimensional conservation equations like the DNS (Direct Numerical Simulation), LES (Large Eddy Simulation) and RANS (Reynolds Averaged Navier Stokes) simulations and numerical methods that use only 1 dimensional conservation equations like subchannel codes. At the top of figure 1, the DNS simulation is shown. It resolves even the smallest scales of the turbulence within a fluid flow and doesn’t need any modelling of physical effects, but it needs the highest computational resources. The next CFD method in the figure is the LES simulation, that resolves the larger turbulence scales directly and has models for the smaller turbulence scales. The RANS simulations uses models for all the turbulent length scales. These three models are all based on the three dimensional conservation equations. In contrast to these methods, the subchannel codes uses only one conservation equation and needs the smallest computational resources, but many physical details have to be modeled. In case of CFD simulations within a reactor core, DNS simulation are not feasible for almost every investigation because of the large geometries and the high Reynolds numbers within the reactor core. LES simulations are possible for parts of a rod bundle, for example a section of a rod bundle containing a spacer. With High-Performance Computing (HPC) simulations and very big supercomputers like for example the Blue Gene, it is even possible to perform LES simulations of a fuel assembly with billions of cells for the computational mesh. RANS simulations are the most common used CFD simulations, but are also not able to simulate a complete reactor core due to computational restraints. For cases where the complete reactor core has to be resolved and analyzed, the use of code coupling to subchannel codes like F-COBRA-TF, NASCA etc. are needed. These subchannel codes use empirical correlations and specific model constants that have to be adapted and adjusted for each new case or geometry. Our new method, the Coarse-Grid-CFD, is situated between the RANS simulation and the subchannel codes. The purpose of the development of this new method is to give an alternative to the traditionally used subchannel codes. The CGCFD should take advantage of desirable features of modern CFD solvers, such as reliability, robustness and graphical user interfaces. Furthermore, the CGCFD should be independent from experimental or empirical input that has to be used in subchannel codes. The CGCFD should computate very large and complex geometries that cannot be solved with “traditional” CFD simulations. CGCFD should profit thereby from the continuous development of modern CFD solvers.

Methodology of the CGCFD

The CGCFD method applies to flow and heat transfer problems where similar flow situations are repeated frequently in a large domain. Instead of simulating the whole domain with many similar flow patterns at distinct locations, the appearing flow patterns are computed once and for all. The global flow is simulated by assembling local behaviors. Such a method works best if the geometry can be subdivided into representative small blocks. Hence, the method of the CGCFD is suitable especially for core wide simulations within a nuclear power plant.
because the flow situation is repeating umpteen times in different rod bundles inside the core. Figure 2 describes the general approach of the methodology of the CGCFD.

The CGCFD method begins with the identification of a representative block within the complete geometry where frequently recurring flow situations appear. Because the selection of the block has a major impact on the results of the CGCFD, this step within the process of the CGCFD is fundamental and requires comprehensive knowledge on the flow situation within the complete geometry. In the next two steps the selected representative block is set up for a detailed CFD-simulation and the detailed CFD simulation is performed. This includes parametric studies for different flow situations that can occur in the geometry. From the results of the detailed CFD so called volumetric forces are extracted. Note that the detailed CFD results will directly enter into the results of the CGCFD. That means that all inaccuracies will enter the results of the CGCFD, So the detailed CFD simulations have to be performed with special care. The method of extraction of the volumetric-force-terms is explained in the following chapter. Since a frictionless Euler solver without turbulence model is used to perform the CGCFD, effects of turbulence and friction are completely embedded in the volumetric forces. In the next step a parameterization of the volumetric-forces is realized so that the detailed CFD-simulation is not necessary for every possible flow condition. For this step one can take advantage of the knowledge of subchannel codes where parameterization of variables is common. Therefore, one will get a robust parameterization for the volumetric forces within the CGCFD. In the last two steps, the coarse mesh for the complete geometry is set up and the CGCFD-simulations are solved using the Euler equations and the implemented volumetric source terms.

### Theory of the CGCFD

In system codes or sub channel analysis the conservation equations are integrated over rather large control volumes and models are used for friction losses and transport between neighboring channels. When using CGCFD the required terms can be evaluated directly from fully resolved fields. The momentum equation from a detailed stationary RANS simulation with a first moment closure reads:

\[
\nabla \cdot (\overline{\rho \bar{v} \otimes \bar{v}}) = -\nabla p + \nabla \cdot \sigma + \nabla \cdot \sigma_t \tag{1}
\]

with the stress tensor \( \sigma \):

\[
\sigma = \mu \cdot \left( \nabla \bar{v} + (\nabla \bar{v})^T - \frac{2}{3} (\nabla \bar{v}) \cdot \sigma_{ij} \right) \tag{2}
\]

and the turbulent stress tensor \( \sigma_t \) that is based on the Boussinesq-Approximation with the turbulent kinetic energy \( k \):

\[
\sigma_t = \mu_t \cdot \left( \nabla \bar{v} + (\nabla \bar{v})^T - \frac{2}{3} \rho k \cdot \sigma_{ij} \right) \tag{3}
\]

For the CGCFD- simulation the frictionless stationary Euler equation is used:

\[
\nabla \cdot (\overline{\rho \bar{v} \otimes \bar{v}}) = -\nabla p
\]

Clearly the use of the Euler equations by themselves does not resolve the subgrid physics like viscous dissipation or turbulence. So the subgrid physical information is taken into account by a volumetric source term \( FC \). The new equation reads:

\[
\nabla \cdot (\overline{\rho \bar{v} \otimes \bar{v}}) = -\nabla p + FC \tag{4}
\]

The stokes integration of equation (1) reads:

\[
\iint (\overline{\rho \bar{v} \otimes \bar{v}}) \cdot \vec{n} \cdot dO = \iint p \cdot \vec{n} \cdot dO + \iint \sigma \cdot \vec{n} \cdot dO + \iint \sigma_t \cdot \vec{n} \cdot dO \tag{6}
\]

and for the equation (5):

\[
\iint (\overline{\rho \bar{v} \otimes \bar{v}}) \cdot \vec{n} \cdot dO = -\iint p \cdot \vec{n} \cdot dO + \iint FC \cdot dV \tag{7}
\]

---

**FIGURE 2. METHODOLOGY OF THE COARSE-GRID-CFD**

- Identify representative block within complete geometry
- Setup of numerical grid for representative block
- Detailed CFD-simulation of representative block
- Post processing and extraction of volume forces employing CFD results
- Parameterization of volumetric force terms using relevant flow conditions
- Setup coarse mesh for complete geometry
- Simulation of complete geometry with coarse mesh employing CGCFD
To obtain the same results as the RANS simulation with the Euler equations that are used for the CGCFD the equations (6) and (7) have to be equal. Therefore we subtract the equation (6) from equation (7). This leads to an equation for the volumetric force term, assuming that $\vec{v}$ and $p$ are almost equal in equations (6) and (7).

$$\iint V \vec{F} C = - \iint \sigma \cdot \vec{n} \cdot dO - \iint \sigma_t \cdot \vec{n} \cdot dO \quad (8)$$

Discretisation of (8) with the corresponding control volume leads to a conditional equation for the volumetric force, that reads:

$$\vec{F} C \cdot V = - \sum_j \sigma \cdot \vec{n}_{Oj} \cdot A_j - \sum_j \sigma_t \cdot \vec{n}_{Oj} \cdot A_j \quad (9)$$

Note that these forces include turbulence and non-resolved form losses. In general, the fully resolved fields are not available. In fuel assembly simulation one typically have the same geometry with near identical flow patterns repeating all over the core region. Therefore, the detailed simulation can be computed for a generic representative section and all possible situations are tabulated or parameterized.

CGCFD OF A WATER ROD BUNDLE EXPERIMENT

To show the functionality of the CGCFD-method we used the results of a water rod bundle experiment that was performed in the KALLA laboratory at the Karlsruhe Institute of Technology.

Experimental set up water rod bundle experiment

Figure 3 shows the water loop with the installed rod bundles and the spacer positions. Table 1 summarizes the technical data of the rod bundle and the flow conditions of the experiment. The water experiment accompanies a geometrically identical HLM experiment with similar integral pressure instrumentation but lacking the optical access. Exhaustive experimental results and a summary of all performed experiments can be found in [5].

RANS CFD-simulation of the water rod bundle experiment

The water rod bundle experiments were accompanied by comprehensive CFD-investigations that were performed at the Institute of Nuclear and Energy Technologies at the KIT. For the validation of the CGCFD several RANS simulations from the detailed CFD investigation of the water rod bundle experiment were used. The first detailed CFD simulation was a half symmetrical rod bundle that includes a flow conditioner, a pin fixer and a spacer grid. The second simulation used a flow domain of a 60° sector of the rod bundle containing 2 spacer grids and the last geometry that was simulated corresponds to the 60° rod bundle, but contains only one spacer grid. Within these CFD investigation we observed that the pressure drop across both spacer grids and the pin fixer are nearly identical so that we only used a single spacer grid as representative section for the CGCFD simulation of the rod bundle. For more details of the simulations of the other geometries see [2], [3].

Figure 4 shows the axial pressure along selected lines through the rod bundle from the RANS simulations that where performed with the 60° computational domain in OpenFOAM. The position of the lines are marked within the cross section of the water rod bundle inside the figure. The numerical parameters can be found in table 2. For the CGCFD, we used this simulation to extract the volumetric forces.

The steady state simulations were performed with Star-CCM and OpenFOAM using the high-Reynolds-number k-ε-turbulence model (Lauder et al.) with a second order convection scheme. The k-ε-number turbulence model was chosen since the pressure drop in the selected rod bundle geometry primarily is due to form losses, see [2], [3]. In subchannels secondary flow may lead to additional friction that cannot be described with the isotropic k-ε-turbulence model. However, in the considered bundle form losses dominate over the friction losses allowing for the use of isentropic models with reasonable accuracy. Moreover, for the purpose of demonstrating the CGCFD consistency between the
<table>
<thead>
<tr>
<th>Geometry</th>
<th>hexagonal</th>
</tr>
</thead>
<tbody>
<tr>
<td>Total Power</td>
<td>-</td>
</tr>
<tr>
<td>Fuel Pins</td>
<td>19</td>
</tr>
<tr>
<td>Pin Diameter</td>
<td>8.2 mm</td>
</tr>
<tr>
<td>Pitch</td>
<td>11.48 mm</td>
</tr>
<tr>
<td>Pin length</td>
<td>1272 mm</td>
</tr>
<tr>
<td>Active height</td>
<td>870 mm</td>
</tr>
<tr>
<td>Grid spacer</td>
<td>3</td>
</tr>
<tr>
<td>Mean velocity</td>
<td>10 m/s</td>
</tr>
<tr>
<td>Mass flow</td>
<td>$\sim$ 13 Kg/s</td>
</tr>
<tr>
<td>Max. heat flux</td>
<td>-</td>
</tr>
<tr>
<td>Inlet temp.</td>
<td>25 °C</td>
</tr>
<tr>
<td>Outlet temp.</td>
<td>25 °C</td>
</tr>
</tbody>
</table>

two numerical methods is more important than the very best accuracy of the reference simulation.

**Numerical set up of the CGCFD of the water rod bundle experiment**

In our previous study of a quadratic wire-wrap fuel assembly [1], it was shown that the coarse grid must at least have five control volumes within each sub channel. The center cell contains most of the axial flow, while the transverse transport between neighboring sub channels is concentrated in the four surrounding cells. For the CGCFD of the water rod bundle experiment we followed the same strategy in the hexagonal assembly where a wedge shaped central cell is surrounded by three hexahedral cells representing each sub channel. In figure 5, the front section of the detailed CFD simulation and the mesh for the CGCFD are shown. We used 33 cells in the 60° cross section and 148 cells in the axial direction. The spacer structure is situated in the subchannels and splits the flow into split streams (compare figure 6 a)). In the CGCFD simulation these details are ignored and the separate flow streams are combined. The flow obstruction of the spacers is modeled in the coarse mesh by deforming the rods so that the flow becomes obstructed by the locally thickened rods as depicted in figure 6 b). The spacer geometry generates a flow obstruction of approximately 26% compared to the cross section without spacer. Note, that the flow domain volume and flow cross sections are preserved. Figure 7 and table 2 shows the numerical set up and the flow conditions for the CGCFD.

**Results of the CGCFD of the water rod bundle experiment**

In figure 8 the results of the CGCFD are shown. Within a first simulation run we were not able to reproduce the correct pressure drop within the rod bundle. Figure 8 depicts the pressure drop of four selected lines through the rod bundle of the detailed RANS simulation and the CGCFD. The position of these lines are marked with green dot labels in the cross section of the rod bundle within the figure. The complete pressure drop was underestimated, (compare figure 8). Nevertheless, the pressure characteristics of the rod bundle were reproduced. Comparing the pres-
pressure gradient excluding the spacer, one can see that the results from the detailed CFD-simulation were reproduced. However, the CGCFD results in the spacer region of the geometry underestimated the pressure drop of the real geometry. The underestimation of the pressure drop in the spacer region is due to very coarse mesh in the spacer region. If figure 6 is compared it is clear that we are not able to reproduce the separation of the flow in separate flow domains and the exact geometry of the spacer with only 33 cells in a cross section of the CGCFD mesh. However to obtain the correct pressure drop, the volumetric forces, by themselves, without an accurate geometry are not able to reproduce the correct pressure drop. To improve the quality of the results of the CGCFD an empirical factor was introduced to adjust the volumetric forces in the spacer region. The volumetric forces within the rest of the geometry were not adjusted. The results of the CGCFD with the adjusted volumetric force terms are shown in figure 9 that depicts the pressure drop along the same four lines like in figure 8. The frictional losses along the rods, pressure drop due to acceleration in the spacer region, and pressure recovery are reproduced on the coarse grid with the adjusted volumetric force term. The CGCFD simulations run three orders of magnitude faster than the detailed simulations. Moreover, it requires much smaller computational resources (Compare the CPU-Time and number of cells in table 2). Note that the CGCFD simulations completely ignore the frictional losses along the rods. As mentioned in the beginning of the paper, the goal is to develop a general methodology that doesn’t need any empirical or experimental input. We seek for a method that is able to reproduce the influence of any geometrical details of a geometry in combination with very coarse meshes. This led to the idea to combine the CGCFD with the anisotropic porosity formulation that was originally implemented in the COMMIX code [6].

**ANISOTROPIC POROSITY FORMULATION AND THE CGCFD**

The anisotropic porosity formulation that we combined with the CGCFD was originally used in the COMMIX [6] code. The code was designed to compute complex multidimensional thermal-hydraulic simulations in a time when computational resources were limited. Therefore the COMMIX code used a anisotropic porosity formulation with the parameters of volume porosity, anisotropic directional surface permeability and a distributed resistance to model complex fluid-dynamic phenomena of internal solid structures.

**Theory of the CGCFD and the anisotropic porosity formulation**

The anisotropic porosity formulation in the COMMIX code is based on three parameters: The volume porosity, the directional surface permeability and a distributed resistance. The volume porosity is defined as the fraction of the local infinitesimal volume element $dV$ that is occupied by the fluid:

$$\gamma = \frac{V_F}{V_C}$$  \hspace{1cm} (10)
Where $V_F$ is the part of the volume element that is unobstructed and $V_C$ is the corresponding complete volume element. The range of values, that $\gamma$ may have lies between zero and unity. Zero means that the volume element is totally obstructed by a solid structure, where unity means that the volume element is totally unobstructed to fluid flow. The volume porosity is a constant for a given control volume. The surface permeability is defined as the ratio of the individual control surface that is unobstructed to fluid flow compared to the corresponding complete control surface.

$$\gamma = \frac{A_F}{A_C}$$ (11)

Where $A_F$ is the individual control surface that is unobstructed to fluid flow and $A_C$ is the corresponding complete con-
The control surface. The values of \( \gamma \) also lie between zero and unity. Zero means that the considered surface is completely impermeable. Unity means that the surface is unobstructed. The surface permeability depends only on geometrical information of the geometry that is under consideration. These two parameters and the distributed resistance term led to the Navier-Stokes-Equation with the anisotropic porosity formulation included that reads for the conservation of mass:

\[
0 = \gamma \nabla \cdot (\rho \vec{v})
\]

and for the conservation of the momentum:

\[
\gamma \nabla \cdot (\rho \vec{v}) = -\gamma \nabla p + \nabla \cdot \sigma \gamma - \gamma R_i
\]

This set of equations is implemented in the COMMIX code. For the CGCFD code, the implementation of the anisotropic porosity formulation reads for the conservation of mass:

\[
0 = \gamma \nabla \cdot (\rho \vec{v})
\]

and for the conservation of momentum:

\[
\gamma \nabla \cdot (\rho \vec{v}) = -\gamma \nabla p + \nabla \cdot \sigma \gamma - \gamma R_i + F \gamma
\]

For the CGCFD we used a steady state solver so the terms \( I_I \) and \( H_I \) can be neglected. The diffusive terms \( H_N \) are resolved by the volumetric source terms that are derived from the fully resolved CFD simulations. Also, the distributed resistance term may be neglected and instead be resolved through the volumetric source terms. This is advantageous because the friction factor is normally not very well known in thermal-hydraulic problems and must be derived from comprehensive experiments. This formulation allows us to use a solver without any empirical or experimental input.

**Setup of the simulation with CGCFD and the anisotropic porosity formulation**

The mesh for the CGCFD consists, like in the previous simulation, without the anisotropic porosity formulation of 33 cells per cross section. In the spacer region we slightly increase the actual resolution in the flow direction to map the geometrical information of the spacer to the coarse mesh with the volume porosity and the surface permeabilities. Figure 10 is a comparison of the old spacer geometry and the new mesh for the CGCFD with the anisotropic porosity formulation. In Figure 10 a) one can see the geometry from the previous simulation, where some material is put on the rods in the spacer region in combination with the volumetric force terms and the factor in the spacer region to get good results. Figure 10 b) shows the new unobstructed mesh in the spacer region where the volume porosity and the surface permeability factors are used to rebuild the effect of the blockage of the spacer. Figure 11 shows a cross section of these spacer geometries with the values of the volume porosity and the surface permeability that are introduced into the finite volume scheme and that are calculated with the equations 10 and 11 from CAD files and the positions and volumes of the corresponding control meshes. Figure 12 and table 3 shows the flow and boundary condition of the water rod bundle for the CGCFD.

![Comparison of mesh spacer region CGCFD and CGFD with porous media formulation](image1)

**Table 3. Numerical set up CGCFD with porous media formulation for water rod bundle experiment**

<table>
<thead>
<tr>
<th>Solver</th>
<th>Inflow</th>
<th>Turb. model</th>
<th>No. of Cells</th>
<th>CPU-Time</th>
</tr>
</thead>
<tbody>
<tr>
<td>Detailed CFD</td>
<td>U=10 m/s</td>
<td>K-( \varepsilon )</td>
<td>500000</td>
<td>ca. 30 h</td>
</tr>
<tr>
<td>CGCFD+PM</td>
<td>U=10 m/s</td>
<td>-</td>
<td>5214</td>
<td>ca. 5 min</td>
</tr>
</tbody>
</table>
FIGURE 11. CROSS SECTION WITH VALUES FOR THE SURFACE PERMEABILITIES AND VOLUME POROSITIES IN THE SPACER REGION

<table>
<thead>
<tr>
<th>Cell No.</th>
<th>xx</th>
<th>yy</th>
<th>xx</th>
<th>yy</th>
<th>xx</th>
<th>yy</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>6</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>7</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>8</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>9</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>10</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>11</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>12</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>13</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>14</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>15</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>16</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>17</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>18</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>19</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>20</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>21</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>22</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>23</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>24</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>25</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>26</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>27</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>28</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>29</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>30</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>31</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>32</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>33</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>34</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>35</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>36</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
</tbody>
</table>

FIGURE 12. BOUNDARY AND FLOW CONDITIONS FOR THE CGCFD WITH POROUS MEDIA FORMULATION

Results of the simulation with CGCFD and the anisotropic porosity formulation

The results of the CGCFD in combination with the anisotropic porosity formulation are shown in figure 13. Compared to the results of the CGCFD without the anisotropic porosity formulation, these results are purely based on geometrical factors such as volume porosity and surface permeability. No empirical factor is used within the spacer region nor the rest of the geometry. Figure 13 shows the pressure drop along 4 lines through the rod bundle. The lines that are marked as detailed are the corresponding highly resolved RANS simulation results and the lines that are marked as coarse are the CGCFD simulations with the anisotropic porosity formulation. Similar as in the previous discussion, the position of these lines is marked in the cross-section within the figure. The CGCFD with the anisotropic porosity formulation reproduces the frictional losses along the rods and the spacer, pressure drop due to acceleration in the spacer region, and pressure recovery with the very coarse mesh.
CONCLUSION

The results that we gained with our new technique, the Coarse-Grid-CFD, that uses highly resolved simulations on representative blocks to parameterize sub-grid forces, well agree with the corresponding detailed CFD simulations. The Coarse-Grid-CFD is a new methodology to compute the flow and predict the pressure field within a fuel assembly with minimal numerical effort. In a first attempt, we produced very promising results and showed that the CGCFD simulation is able to reproduce results from detailed CFD simulations with considerably larger meshes. Nevertheless we still needed a factor in the spacer region to reproduce the correct results. To be a true alternative to the sub-channel codes the CGCFD has to differ from them. This means that the use of empirical or experimental input has to be avoided. This constraint led to the idea to combine the CGCFD with the anisotropic porosity formulation that was originally implemented in the COMMIX code. The results that were produced with the CGCFD in combination with the anisotropic porosity formulation well reproduces the results from the corresponding detailed CFD simulation.

The advantage of the Coarse-Grid-CFD, especially of the CGCFD with the anisotropic porosity formulation, compared to the state of the art subchannel analyses, is that no tuning parameter, nor empirical or experimental input is needed to adjust the solvers for a specific geometry. Certainly, this method requires user dedicated suitable parametric dependency, and it requires educated user input to identify the representative blocks that are needed to extract the subgrid forces. Finally, the detailed CFD simulations, whether they are RANS or LES simulations, need models which require careful use, but can be validated for general classes of applications. For the use of these models there are often best practice guidelines proposed. Since similar flow conditions repeat umpteen times, the costs of the representative CFD simulations that are needed to extract the volumetric forces is much lower than a full simulation. Due to the coarse meshes there is the possibility to use this method to compute a full reactor core and the adjacent plena in a single CFD simulation. Geometry variations such as obstructions can be considered with the CGCFD and the anisotropic porosity formulation. By implementing the CGCFD into standard CFD software the reliability and robustness of modern solvers can be taken into account.

ACKNOWLEDGMENT

This work is supported in the framework of the FP6-IP-EUROTRANS project; Contract number FI6WCT-2004-516520, FP7-THINS [8], the AREVA Nuclear Professional School (ANPP) and the Karlsruhe House of Young Scientists (KHYS).

REFERENCES