A CFD APPROACH TO THE ATMOSPHERIC DISPERSION OF RADIONUCLIDES IN THE VICINITY OF NPPS

Paulo A B de Sampaio¹, Milton A G Junior² and Celso M F Lapa³

Instituto de Engenharia Nuclear (IEN / CNEN - RJ)
Caixa Postal 68550
21945-970 Rio de Janeiro, RJ
¹sampaio@ien.gov.br
²miago@ien.gov.br
³lapa@ien.gov.br

ABSTRACT

Most studies of atmospheric dispersion of radionuclides released from Nuclear Power Plants (NPPs) are based on Gaussian plume models or on the use of a convection-diffusion equation. Such methods, which do not involve solving the flow problem, are useful in the atmospheric mesoscale, of the order of 2-2000 km from the NPP. However, they do not account for the turbulence generated by the interaction of the wind with obstacles and with the released material stream, which are the dominant factors in the local scale, of the order of 0-2 km from the source of emission. In order to study the dispersion of radionuclides in the vicinity of NPPs, the authors advocate the use of Computational Fluid Dynamics (CFD). The physical model is based on the Navier-Stokes equations, a convection-diffusion energy equation, and transport equations for the radionuclides. The stabilized finite element formulation employed results in a Large Eddy Simulation procedure, where no explicit subgrid modeling is required. The code uses adaptive techniques combining error estimation and remeshing. It has been implemented in a Beowulf parallel computing system using domain decomposition and the Message Passing Interface (MPI) for communication among processors. Both controlled emissions from a chimney and severe accidents have been simulated, showing the importance of the local phenomena on the dispersion of radionuclides.

1. INTRODUCTION

In this paper the authors propose the use of Computational Fluid Dynamics (CFD) to the problem of atmospheric dispersion of radioactive material in the vicinity of NPPs. Most studies of atmospheric dispersion of radioactive material released from NPPs are based on Gaussian plume models or on the use of a convection-diffusion equation. Although they are useful in the atmospheric mesoscale [1], these models fail to take into account the turbulence generated by the interaction of the wind with buildings, the terrain, and with the convective forces of the released material stream, which are the dominant factors in the local scale, of the order of 0-2 km from the source of emission.

The CFD approach proposed herein employs a stabilized finite element formulation to approximate the solution of a physical model that is based on the Navier-Stokes equations, coupled with an energy convection-diffusion equation for temperature, and transport equations for the concentration of the released radionuclides. The stabilized finite element
2. PHYSICAL MODEL, DISCRETIZATION AND CODE IMPLEMENTATION

The flow is modeled by the incompressible Navier-Stokes equations, a convection-diffusion energy equation and two concentration-transport equations, which represent a first radionuclide that is assumed to decay forming a second one. The dependent variables are the velocity, pressure, temperature and concentration fields. The fluid is considered as Newtonian. The heat flux and diffusive fluxes of concentrations are given by Fick’s law, being the model completed by boundary conditions and initial velocity, temperature and concentration fields. A detailed presentation can be seen in [12-13].

The formulation is based on ideas presented in [9] and [10], but here the method was extended to address the transport of radionuclides. The approach involves the discretization of the governing equations with respect to time and space, followed by minimization of squared residuals with respect to the degrees of freedom. The resulting finite element weighting functions have the same structure of the SUPG weighting of Hughes and Brooks [11] and the stabilization terms emerge naturally from the least-squares minimization procedure, as shown in [12-13]. The stabilized finite element formulation can be interpreted as a Variational Multiscale (VMS) method [2-5] whose application results in Large Eddy Simulations (LES) with implicit subgrid modeling [6-8].

At present we have developed a computer code for the study of 2D problems, where linear triangles are used to interpolate all dependent variables. Even in 2D, the analysis of dispersion of radionuclides with the present method is very computationally intensive. In order to make the method viable to solving practical problems, we make use of dynamic adaptive finite element meshes, based on error estimation, and parallel programming optimized for high performance on a Beowulf-type parallel computing system.

3. NUMERICAL SIMULATIONS

In both examples presented here, we consider the first radionuclide to be Iodine 136 and the second to be Xenon 136. Their concentrations are denoted by $\phi$ and $\varphi$, respectively. In the first example we consider the case of a controlled emission from a chimney and we investigate the effect of the emission temperature and of an irregular topography on the radionuclides concentration levels. We assumed that the temperature of the material released is 100 K higher than the atmospheric temperature. The wind is blowing from left to right at the speed of 2 m/s. In this example we have the interference of the flow with the irregular terrain combined with intense buoyancy forces that lift the released material upwards. This case depicts a severe accident where the plant operators have been forced to release some hot radioactive material to the atmosphere in order to control the internal pressure in the reactor building. In Figure 1 the velocity modulus on the plume vary from 1.6 to 3.8 times the reference velocity $u_0$. Figure 2 depicts adaptive meshes and concentration fields for the same problem.
Figure 1. Isolines of velocity modulus after 10 minutes: only values higher than $1.6u_0$ are shown.

Figure 2. Results during the transient: adaptive meshes (first column), concentration $\phi$ (second column) and concentration $\varphi$ (third column).
In our second example, we address a catastrophic event, where we postulate the failure of all barriers, including the rupture of the reactor building walls. In this example, plume behavior is strongly dependent on the Richardson number (Ri), which expresses the relative strength between free and forced convection. The transient is generated by unbalanced buoyancy forces that exist at the initial conditions assumed, and by the presence of postulated holes in the reactor building walls. At the initial conditions we consider that the reactor building has an internal temperature 300 K higher than the outside temperature. Figures 3 and 4 present numerical results. The wind is blowing from left to right at the speed of 1 m/s for the case of Ri=490, and at the speed of 10 m/s for the case of Ri=4.9. Figure 3 shows isolines of the velocity modulus for case with Ri=490 after 2.5 minutes.

![Image](image.png)

**Figure 3. Isolines of velocity modulus after 2.5 minutes: only values higher than 7 m/s are shown (Richardson number 490).**

Only values higher than 7 m/s are shown in Fig.3. Note that velocities as high as 16.781 m/s (more than 60 km/h) can be observed. The intense buoyancy forces that are active in this example cause the high velocities attained by the plume. Indeed, Fig.3 shows that in a vast region it is the accident that drives the surrounding atmosphere and not the opposite.

### 4. CONCLUDING REMARKS

Most models traditionally used in the study of the dispersion of radionuclides released from NPPs, such as Gaussian Plume Models, do not involve solving the flow equations. Their success in the mesoscale range (2-2000 km) rests on the use of well-tuned parameters that are selected according to predefined weather conditions. However, because they do not model the flow directly, they are not suited to deal with the turbulence that is generated locally from the interaction of the wind with buildings, the terrain and the convective forces of the released material stream. Using our CFD code we have been able to simulate the evolution of the plume as a function of the release temperature, the wind velocity and the size and location of the source of emission.

INAC 2007, Santos, SP, Brazil.
The technical meeting on the *Use of Computational Fluid Dynamics Codes for Safety Analysis of Nuclear Reactor System* held in Pisa in 2002 concluded that CFD codes have a broad potential for qualitative assessment in areas where traditional methods are inadequate [14]. This remark seem to apply to the CFD application proposed in this work, namely, the use of CFD to address the atmospheric dispersion of radionuclides in the vicinity of NPPs.

Although we have considered here only two radionuclides, more complex decay chains can be easily accommodated, with minor modifications in the computer code. The development of a 3D code is also planned.

Besides the application we have focused here, the computer code developed can be applied in the dispersion of radionuclides in other radioactive installations and laboratories such as research reactors, cyclotrons, food irradiation industries, nuclear fuel reprocessing plants, nuclear waste deposits etc.

![Figure 4. On left: Results of $\phi$ - concentration (Richardson number 490): from top to bottom, isolines at $t=50.6$ s, $t=99.4$ s and $t=150.0$ s. On right: Results of $\phi$ - concentration (Richardson number 4.9): from top to bottom, isolines at $t=67.5$ s, $t=132.5$ s and $t=200.0$ s.](image)
REFERENCES