APPLICATION BRIEFS FROM FLUENT

EX158

Aerosol Deposition in a Condenser

In this example, condensation in an emergency heat exchanger used in a nuclear power plant is modeled using FLUENT. Based on the flow field results, the deposition of aerosol particles in the near wall region onto the tube walls is simulated. These particles are generated during a severe core melt accident, and their deposition onto the condenser walls could impact heat transfer. The analysis has helped engineers understand the mechanisms that are most important in the deposition process.

Aerosol particles are likely to be generated during a severe core melt accident at a nuclear power plant. If these particles strike and attach to the walls of the heat exchanger, they can impair heat transfer and lead to potentially high temperatures and pressures. A simulation designed to understand the deposition process is described in this example. It is part of the on-going AIDA (Aerosol Impact and Deposition Analysis) project at VTT Energy in Finland. The results of this analysis are relevant to the workings of a full scale PCC (Passive Containment Condenser) of an ESBWR (European Simplified Boiling Water Reactor) plant. Passive safety systems such as the PCC are gaining popularity in nuclear plant design because they rely more on natural circulation and less on active components, such as pumps and valves, in the reactor control system.

The simulated device consists of concentric pipes (tubes) oriented vertically with an annular space between them. A hot vapor-nitrogen mixture enters the inner tube from above and is cooled by a counterflow of water in the annular region. The cooling process causes some of the vapor to condense on the inner tube wall. Aerosol particles are assumed to be present in the gas flow. They have very little inertia, so their presence does not impact the flow, but their behavior can be deduced from the converged gas flow results.

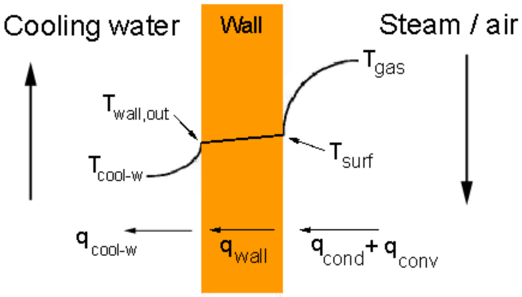

Figure 1: Schematic of the heat transfer in a heat exchanger tube

In 1D, severe accident modeling codes that are primarily used for this purpose (e.g. MELCOR, MAAP and VICTORIA), it is typically assumed that aerosol particles and vapor species are evenly distributed throughout the flow. In realistic flows important to reactor studies, this is usually not the case because of large nonuniformities in the velocity, species concentration, and temperature profiles. Aerosol deposition and gas condensation rates must therefore be modeled in two or three dimensions for accurate predictions. To meet this goal, FLUENT, with additional steam condensation and aerosol deposition subroutines, has been used.

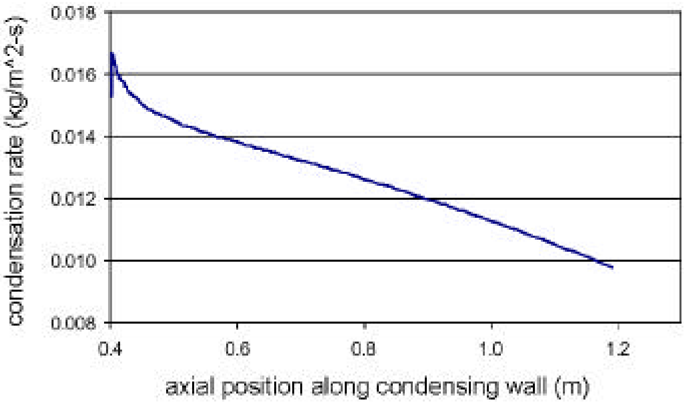

Figure 2: Vapor condensation rate as a function of position along the tube wall

The CFD model consists of the gas-vapor flow space and the surrounding tube wall. The temperature boundary condition on the tube wall is updated with each flow field iteration, based on a one-dimensional heat balance calculation at the wall. (See Figure 1.) The surface where condensation occurs is in

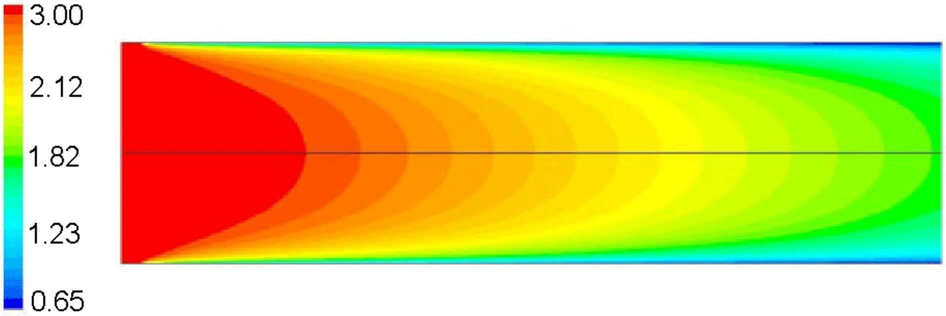

Figure 3: Contours of the ratio of vapor to nitrogen volume fraction in the heat exchanger tube

thermodynamic equilibrium at saturated conditions. A subroutine solving for the surface temperature and the condensed vapor mass is called, and these values become target values to which the wall temperature and sink terms in the species, continuity, and momentum equations are changed. Even though mass is transferred from the gas to the condensate film, and the wall temperature is adjusted because of the presence of the film, the film itself is not modeled.

The rate of condensation along the inner tube wall, computed by FLUENT, is plotted as a function of position in Figure 2. At the beginning of the tube, the condensation rate declines rapidly but then declines at a more uniform rate until the end of the tube is reached. This behavior is also illustrated in Figure 3, where a contour plot of the ratio of vapor to nitrogen volume fraction is shown. Condensation is strongest at the beginning of the tube, where the vapor is in highest concentration (red). Thereafter the vapor volume fraction depletes and a layer of noncondensable gas (nitrogen, blue) starts to build up at the condensing surface. The reduction in the radial concentration gradient of the vapor offers resistance to further diffusion of vapor to the wall and subsequent condensation. The condensed vapor mass flow rate to the wall can be deduced from the difference between the vapor / nitrogen ratios at the inlet and outlet.

A calculation of the particle deposition properties, based on the flow field results, helps identify the mechanisms that are most important to this process. In this simulated case, particle deposition mechanisms are primarily due to concentration and temperature gradients in the continuum flow field (diffusiophoresis and thermophoresis, respectively). These gradients are governed, in part, by condensation on the tube walls. In Figure 4, the particle velocities perpendicular to the primary flow are shown as a function of axial position. The velocities based on diffusiophoresis (blue curve, top) and thermophoresis for a range of particle sizes (multiple colors, bottom) are shown. These results indicate that diffusiophoresis, which is independent of particle size, is the more important particle deposition mechanism when condensation occurs, and that smaller particles impact the walls with higher velocities than larger ones. These velocities can be used to evaluate the deposition properties of the system.

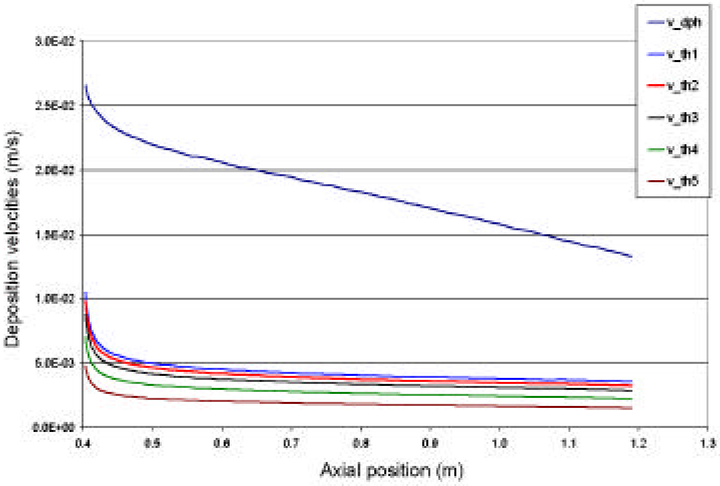

Figure 4: Particle deposition velocities perpendicular to the wall as a function of axial position based on concentration gradients (top) and thermal gradients (bottom)

In this study 36% of the vapor condensed. Without taking condensation into account, the flow conditions would have been calculated incorrectly, since the mass flow at the outlet would have been overestimated by 50%. This would have impacted the temperature and concentration gradients, and thus the particle velocity calculation. It is therefore very important to include the condensation in this type of calculation.

This work is part of the “New technical solutions for the next generation nuclear power plants”s project funded by the Finish Technology Development Center.

Courtesy of Wolfgang Ludwig, Aerosol Technology Group, VTT Energy, Finland