Computational Fluid Dynamics (CFD)

A State-of-the-Art Approach to Safety Analysis

U.S. Nuclear Regulatory Commission

What is CFD?

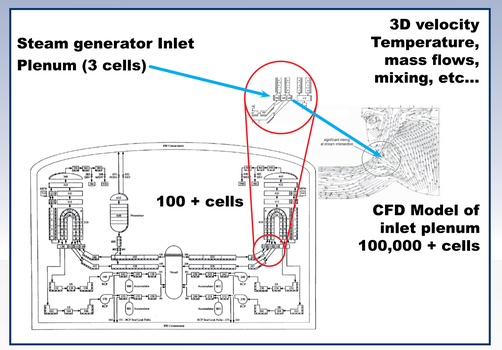
- **C**omputational **F**luid **D**ynamics refers to a numerical solution of the **Reynolds-Averaged Navier-Stokes** equation set.
  - ❖ Conservation of mass, momentum, and energy for a continuous fluid.
  - ❖ Non-linear equations solved numerically on a discrete mesh representing the geometry.
  - ❖ Provides detailed three-dimensional predictions of fluid flow, mixing, and heat transfer.

Why does the NRC use CFD?

- Reactor simulation codes such as RELAP5 or TRACE are basically 1D pipe flow tools most suitable for predicting overall system behavior and component to component interactions (no details).
- CFD provides details that can play a role in …
  - ❖Confirming the robustness of Risk Informed licensing decisions
  - ❖Reducing uncertainty
  - ❖Improving the technical basis for future licensing requests

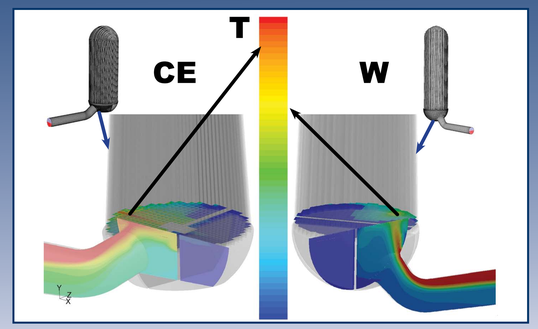
CFD Augments System Code Analysis

- Specific aspects of the system behavior can be focused on using CFD
- System codes can provide boundary conditions for the detailed analysis



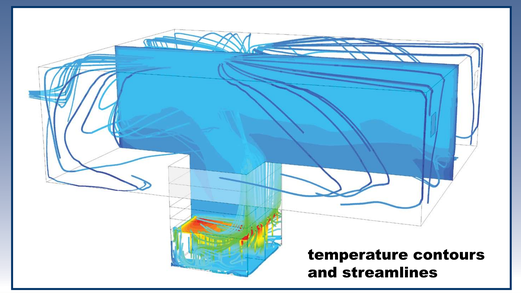
Steam Generator Inlet Plenum Mixing During a Severe Accident

- During a particular severe accident scenario, hot gases from the core circulate through the hot legs and steam generator in a counter current flow pattern. Mixing of the gases in the steam generator inlet plenum significantly reduces the thermal challenge to the thin tubes in the steam generator.
- State-of-the-Art CFD predictions have helped reduce our uncertainty on the inlet plenum mixing and have significantly improved our technical basis for licensing decisions in this area



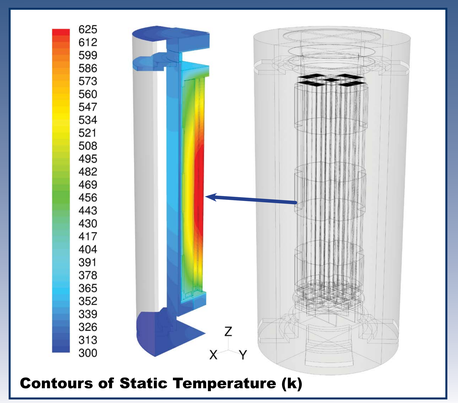
Fuel Pool Accidents Natural Circulation Cooling

- Spent fuel can be air cooled after a complete loss of pool coolant.
- CFD predicts the complex three dimensional flow patterns within the fuel racks and throughout the storage building that can impact the fuel coolability.
- Cooling under a variety of scenarios has been considered.

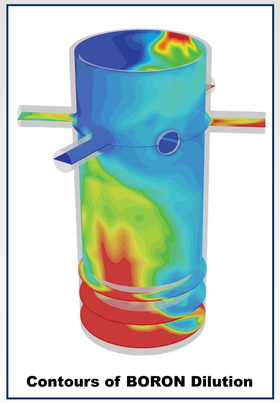


Dry Cask Analysis

- The CFD approach has been used to study a number of cask designs under a variety of external conditions (fires, reduced ventilation, hotter fuels, etc..)
- This work supports dry cask certification efforts and improves our technical bases for licensing decisions.



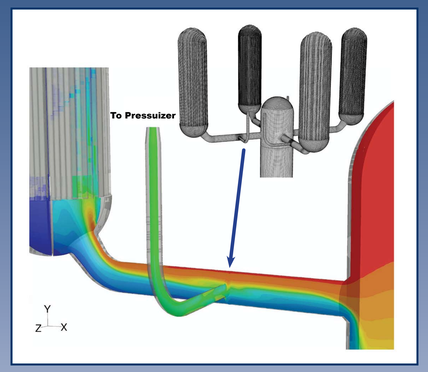
BORON Dilution and Transport Mixing in a Reactor Downcomer



- Condensation in SG tubes during certain loss of coolant accidents can result in the formation of coolant slugs with significantly reduced BORON concentration. These slugs, if unmixed, could lead to a reactivity initiated event when passing through the core region.
- CFD predictions of the complex mixing of the dilute slug as it passes through the downcomer helped reduce our uncertainty of this complex phenomenon.

Hot Leg Counter Current Flows During a severe accident

- During a severe accident scenario in a PWR, flow from the vessel is cooled in the SG and returns to the vessel in a counter current flow pattern.
- The rate of this flow impacts the core cooling and the challenge to the SG tubes.
- This complex three-dimensional flow pattern is predicted using CFD to provide the NRC staff with a significantly improved understanding the flow rate and factors that can influence it.
- The predictions have been used to extend a limited set of test data into full-scale severe accident conditions for a variety of configurations.



Capability in Place and Prepared for the Future

The Office of Nuclear Regulatory Research (RES) has developed a significant "in-house" capability to perform state-of-the-art CFD predictions and this capability already plays a significant role in the RES mission.

The use of CFD is expected to increase in the future with advanced reactor designs and the need to reduce uncertainties. RES staff has developed this capability and is prepared to respond to these future needs.

Office of Nuclear Regulatory Research