



## Safety Enhancement using C(M)FD<sup>1</sup> Analysis

Computational Fluid Dynamics or CFD is the analysis of systems involving fluid flow, heat transfer and

Multi-Phase Thermal-Hydraulics

associated phenomena such as chemical reactions by means of computer-based simulation.

An introduction to Computational Fluid Dynamics – H.K. Versteeg

## Sub Channel Analysis

Support during the design and optimization process

 Identification of potential temperature peaks, sudden pressure losses







## **Direct Contact Condensation Analysis**

Helps to identify regions that in certain flow regimes can lead to potential harmful Condensation Induced Water Hammers
A tool to design successful avoidance criteria







1 – Computational Multi-Fluid Dynamics