# Computational Fluid Dynamics (CFD) Validation Plan

Risk Informed GSI-191 Resolution

Texas A&M University

July 26, 2011

# Outline

- Purpose
- General Approach Description
- CFD Simulations Approach

# Purpose

- To validate and assess the capabilities of CFD code on studying the debris transport and deposition in the reactor core.
- To implement the model to be used for the Downstream Effect Analysis

# General Approach Description

**Phase 1: Data Collection**

Detailed information about existing experimental activities conducted on fuel bundles will be collected. This will includes:

- ✓ Geometry and dimensions of the experimental facility used (drawings). This should include the tested fuel assembly type (e.g. 17x17 or others) and the spacers type.
- ✓ Experimental procedure description.
- ✓ Debris Preparation, Characterization, Quantities, Concentration, and Injection Method.
- ✓ Boundary Conditions (flow rate, pressure, temperature).
- ✓ Experimental Results (pressure, pressure drop, velocities, debris deposition).

**Phase 2: Model Preparation and Testing**

The mesh of the given geometry will be constructed and mesh sensitivity studies will be performed.

**Phase 3: Simulations and Results Analysis**

Simulation results will be compared with the experimental data in order to validate the model.

# CFD Simulations Approach

**General**

- Single Phase Flow Analysis.
- Isothermal Conditions.
- Discrete Phase Method (DPM) approach will be attempted to model the debris inside the reactor.

**Software Candidates:**

- STAR-CCM+
- Ansys CFD

**Modeling:**

- k-ω Shear Stress Transport (SST) Turbulence Model
- Porous Media to model grid spacers and core supports