Fluent/RELAP5-3D Coupling

Analysis of Mixing in VHTR Lower Plenum

David G. Schowalter, PhD
Fluent, Inc

Richard R. Schultz & Walt L. Weaver, PhD
INEEL

The Right Answer in CFD

CFD and the GT-MHR Analysis

- What is CFD?
- CFD in the nuclear industry
- GT-MHR Analysis details

INEEL GENERAL ATOMICS FLUENT The Right Answer in CFD

What is CFD?

- CFD, Computational Fluid Dynamics, is the science of using computational methods to solve physical flow and heat transfer problems.

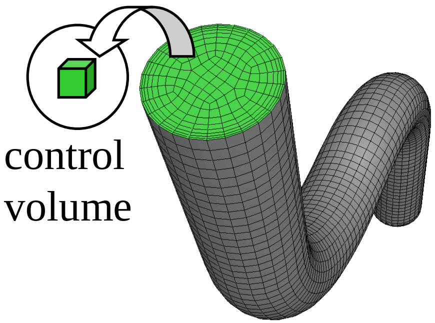
- With advances in computer technology, CFD has become a common tool for equipment design, R&D and process optimization.

What is CFD?

- CFD process entails
  - subdividing the domain into smaller pieces
  - Solving general conservation (transport) equations for mass, momentum, energy, species (gas components), etc.,

$$\underbrace{\frac{\partial}{\partial t}\int_V \rho\phi\, dV}_{\text{unsteady}} + \underbrace{\oint_A \rho\phi \mathbf{V}\cdot d\mathbf{A}}_{\text{convection}} = \underbrace{\oint_A \Gamma\nabla\phi\cdot d\mathbf{A}}_{\text{diffusion}} + \underbrace{\int_V S_\phi\, dV}_{\text{generation}}$$

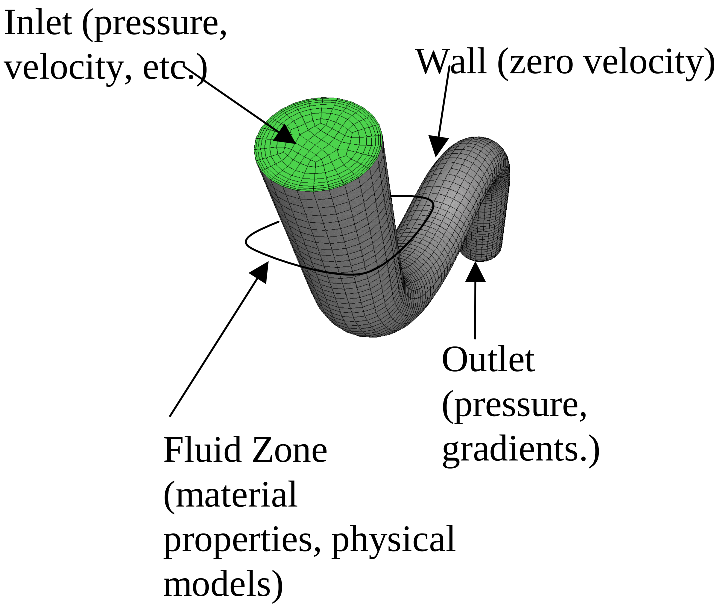
Eqn.	ϕ
continuity	1
x-mom.	u
y-mom.	v
energy	h



Fluid region of pipe flow is discretized into a finite set of control volumes (mesh).

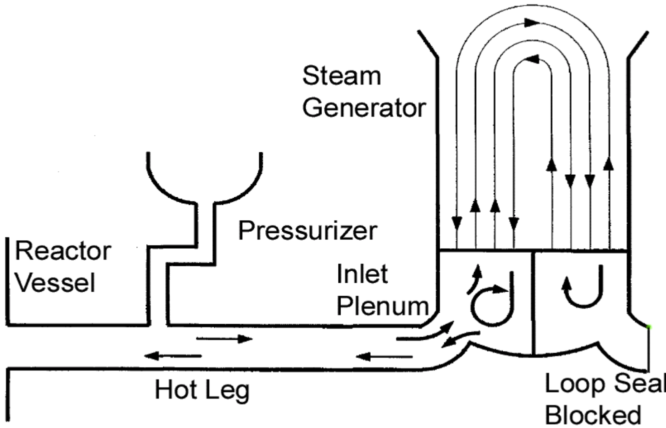
What is CFD? (2)

- A given CFD model contains:
  - Grid/Geometry
  - Physical Models
    - Turbulence
    - Multiphase
    - etc.
  - Boundary Conditions
- Linear equations are solved iteratively



Example: Severe Accident Steam Generator Flows

- FLUENT is used to simulate the flow in a steam generator following a severe loss of coolant accident
- Due to blockage in the coolant loop and seals, cooling is impaired
- The simulation is used to determine the extent of the cooling failure – whether it is restricted to the coolant piping or extends into the steam generator tubing



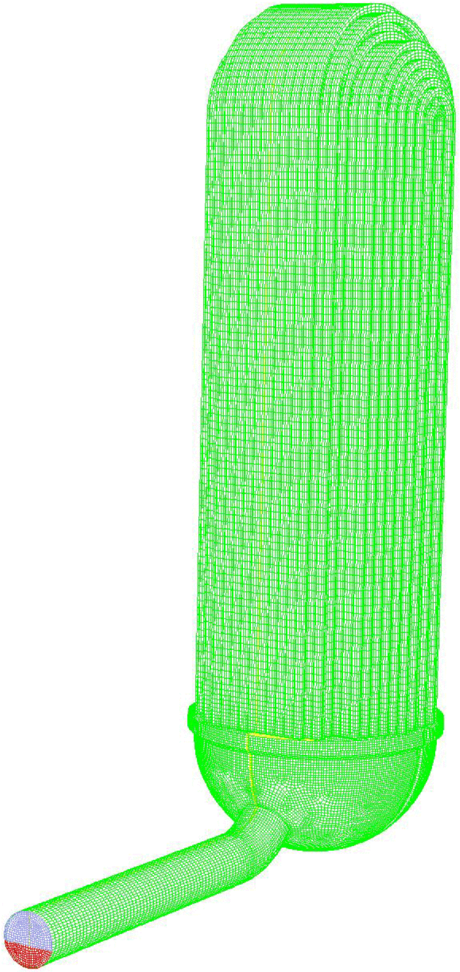
Courtesy of the US Nuclear Regulatory Commission

INEEL GENERAL ATOMICS FLUENT The Right Answer in CFD

Severe Accident Steam Generator Flows

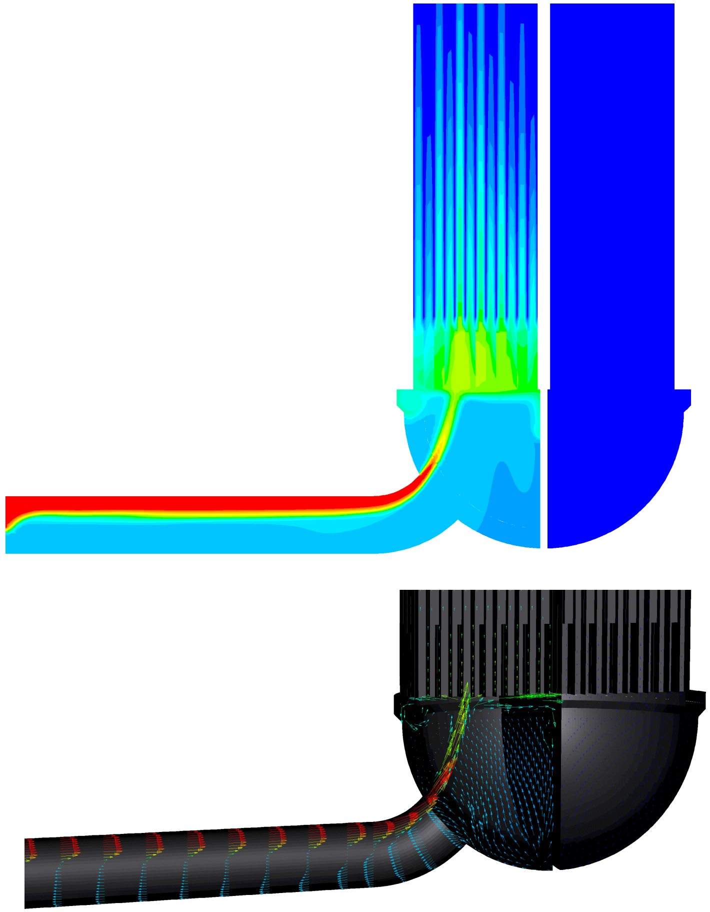
- A mesh of over 500,000 cells is used for the simulation
- The 216 tubes inside the generator are modeled with a square cross section
- A porous media is used inside the tubes
  - This avoids the need to resolve the boundary layer flow in the tubes
- Results are compared to data from a 1/7th scale Westinghouse facility

Courtesy of the US Nuclear Regulatory Commission



Severe Accident Steam Generator Flows

- Temperature contours on the symmetry plane (top) show hot coolant on the top of the pipe
- Velocity vectors (bottom) show the directions of the hot and cool liquid flows
- The predicted average hot and cold temperatures are within 1.5 – 2K of the measured values

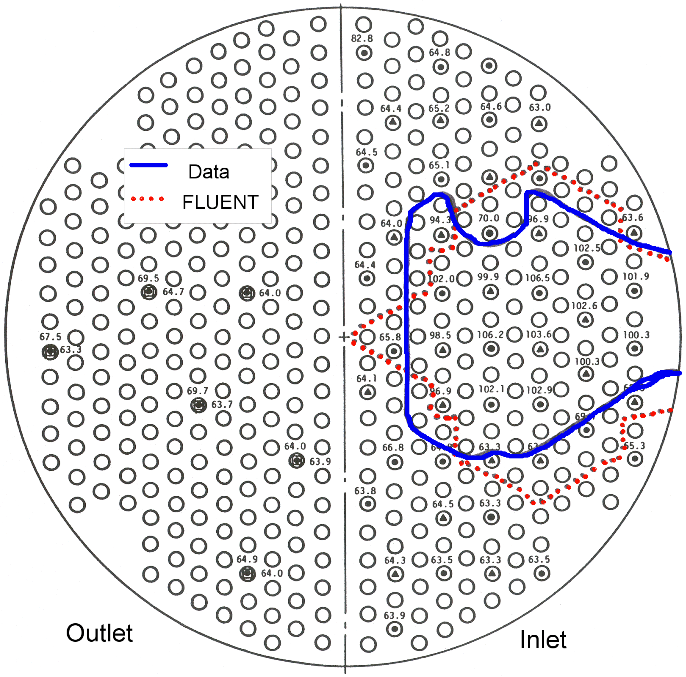


Courtesy of the US Nuclear Regulatory Commission

INEEL GENERAL ATOMICS FLUENT The Right Answer in CFD

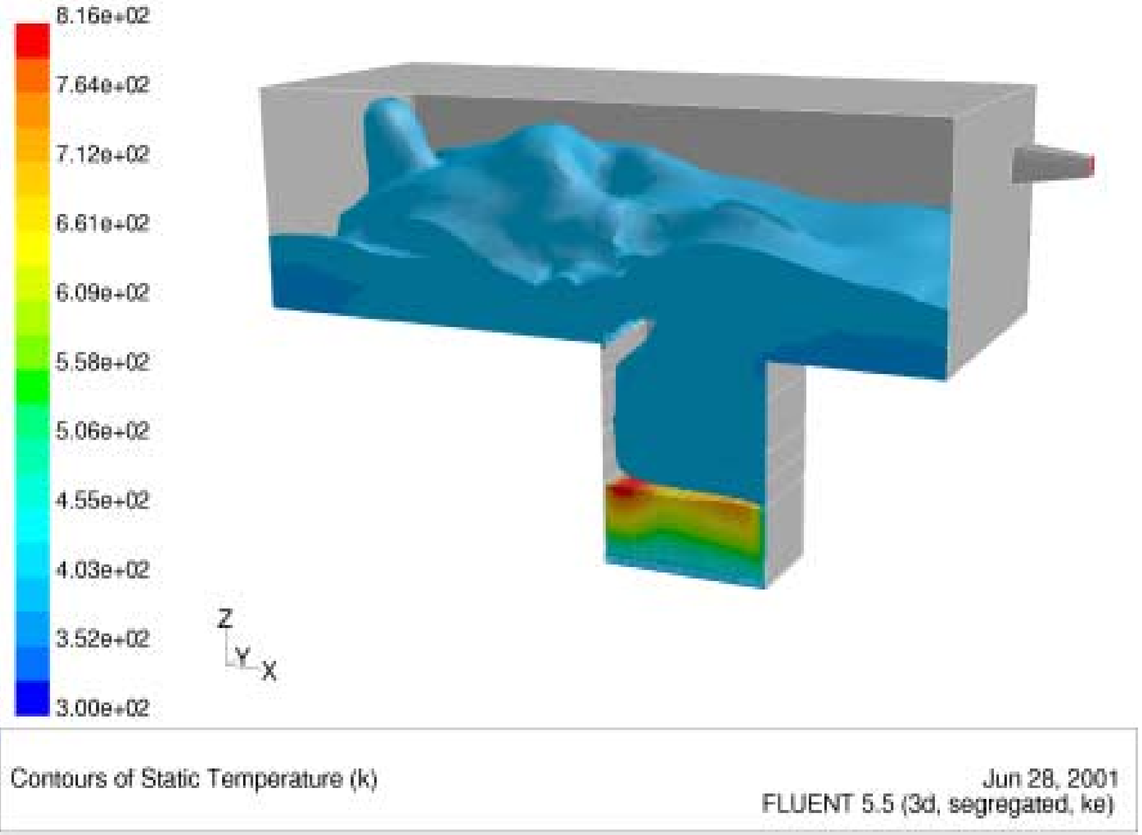
Severe Accident Steam Generator Flows

- Predictions of the number of tubes carrying hot flow are compared to data
- FLUENT over predicts the region of hot flow by 16 tubes, or 7% of the total bundle
- Overall, the CFD results are in very good agreement with data and provide more information than a limited number of thermocouples can



Courtesy of the US Nuclear Regulatory Commission

Example: Pool Storage



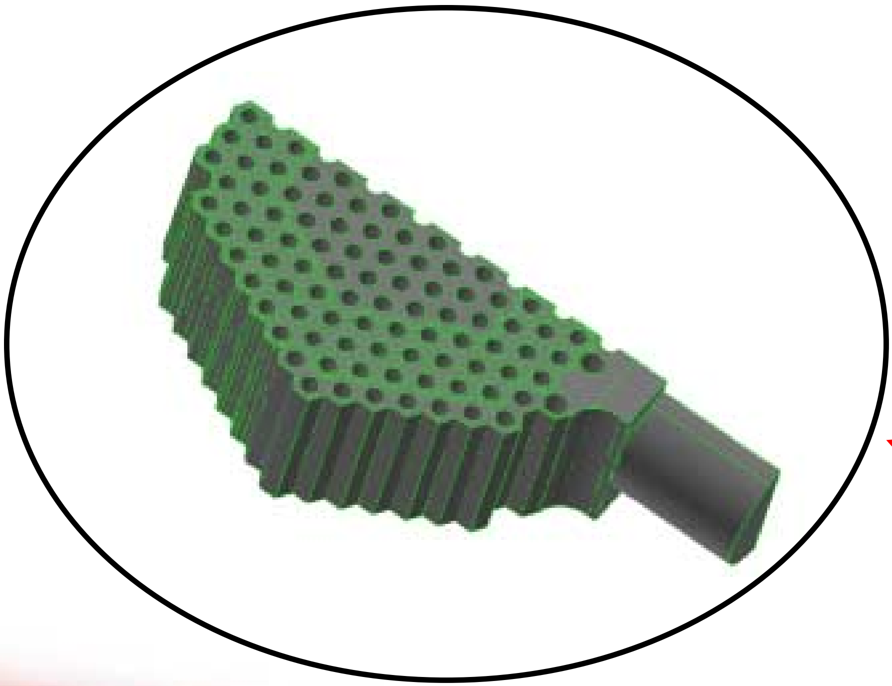
Courtesy of USNRC

- The USNRC used FLUENT to analyze an accident scenario in which there was a loss of water from a storage pool.
- Peak temperatures and cooling patterns were successfully predicted under various scenarios, including different ages of the spent fuel
- The pool is filled to capacity with fuel rods in high density racking.
- Shown is a surface of constant temperature along with contours of temperature in the region of the stored rods.

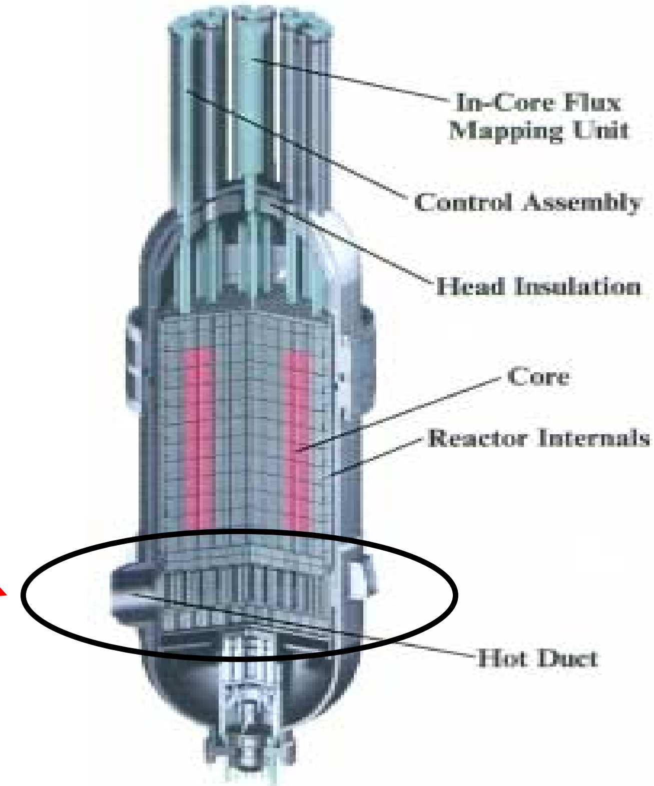
INEEL GENERAL ATOMICS FLUENT The Right Answer in CFD

GT-MHR Lower Plenum Analysis

FLUENT Domain

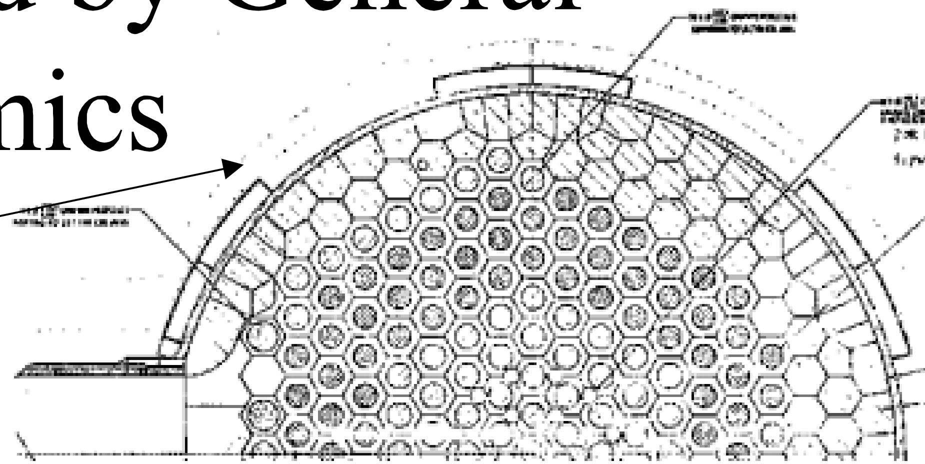


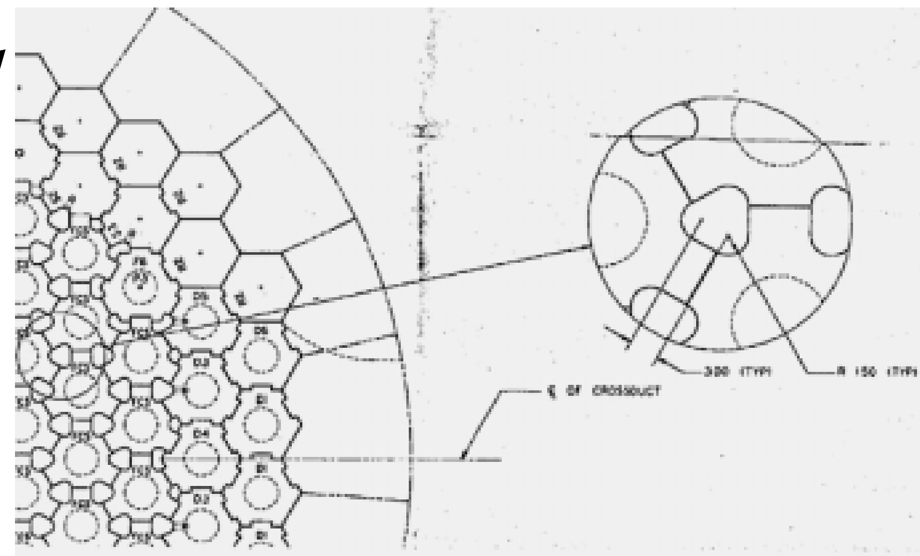
600 MW GT-MHR



Data Provided by General Atomics

- Geometry via drawings
- Output of Sinda model giving core outlet temperatures for each column
- Operating pressure (7 Mpa)
- Flow rate (160 Kg/S helium for ½ model)





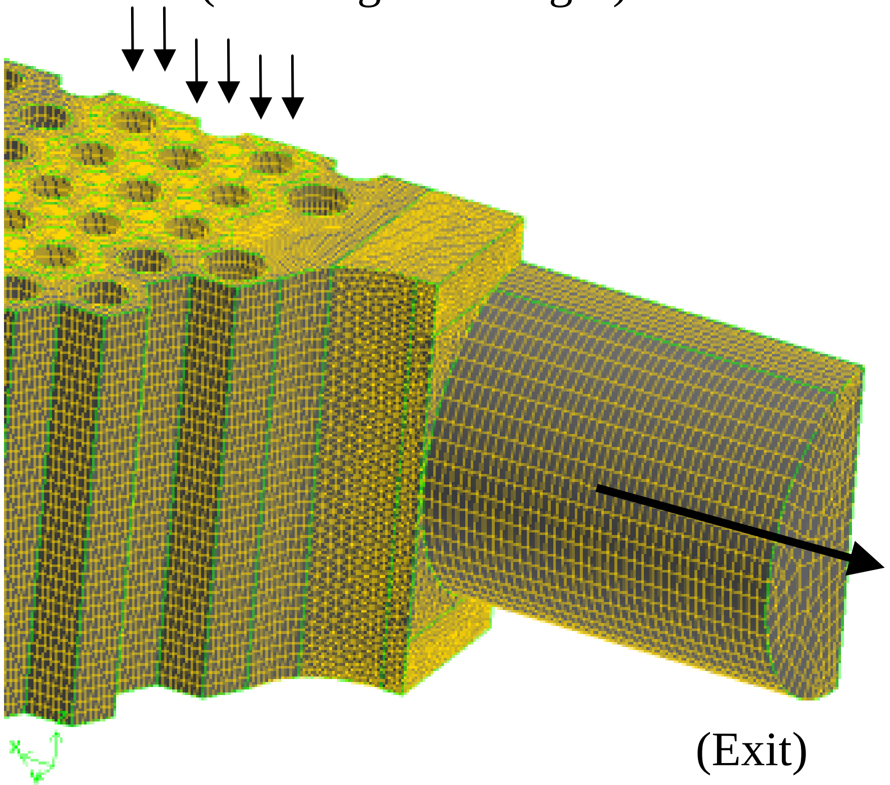
INEEL GENERAL ATOMICS FLUENT

The Right Answer in CFD

Model Details

- ~ 750,000 cell, symmetric mesh
- Buoyancy enabled
- walls treated as adiabatic surfaces
- working fluid is Helium gas
- inlets located around circular pedestals (shown in figure), which support fuel rods
- horizontal nozzle connection provides exit from domain

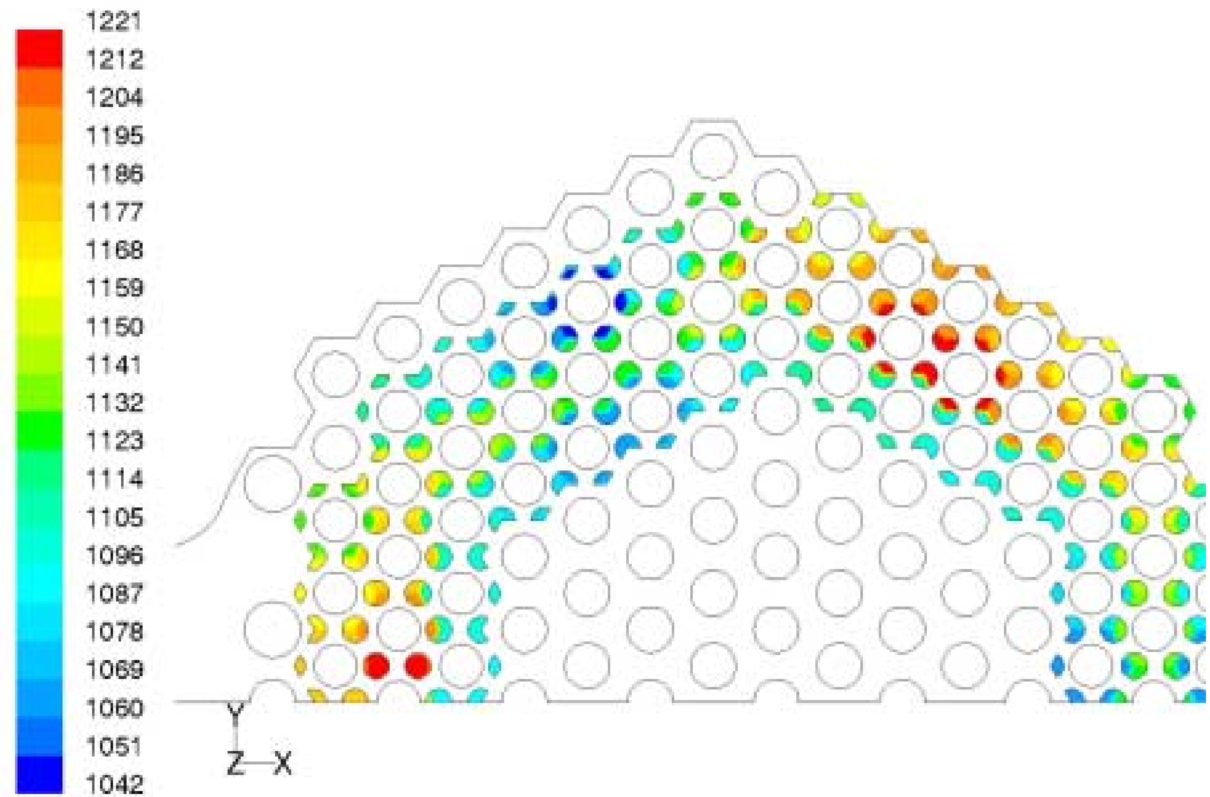
(Entering Helium gas)



(Exit)

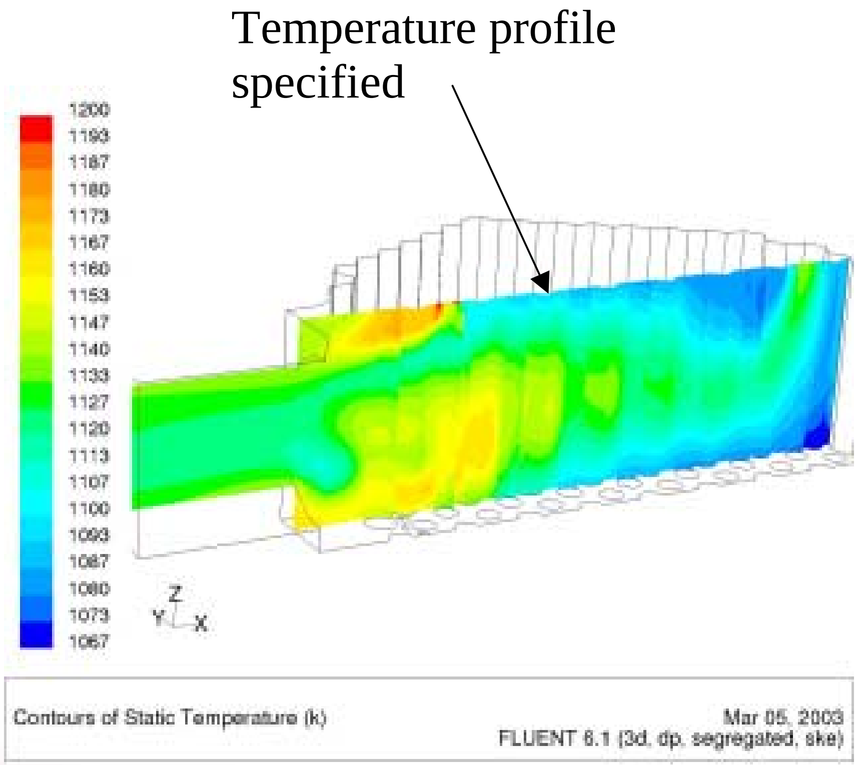
INEEL GENERAL ATOMICS FLUENT The Right Answer in CFD

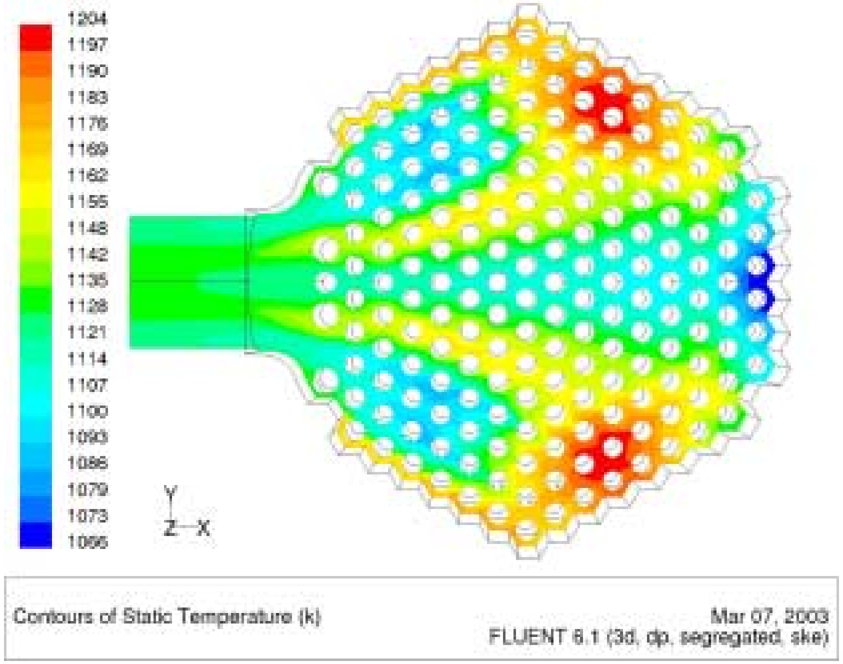
Thermal Boundary Conditions



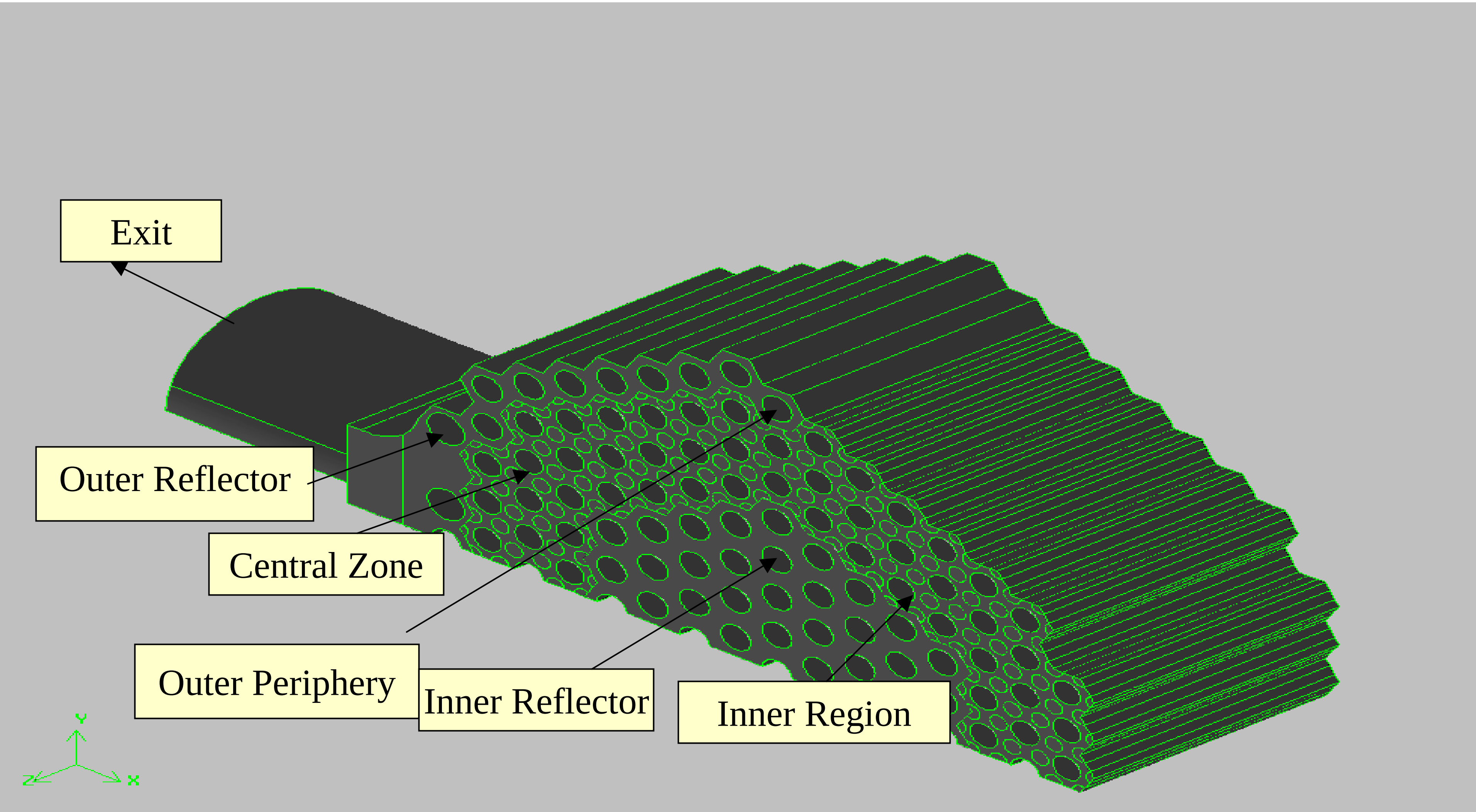
Contours of Temperature Profile Specified at the Velocity Inlets Aug 18, 2003
FLUENT 6.1 (3d, dp, segregated, ske)

GT-MHR Lower Plenum Results:





RELAP5 Coupling: Boundary Conditions Provided at 6 Locations in 1st Calculation…



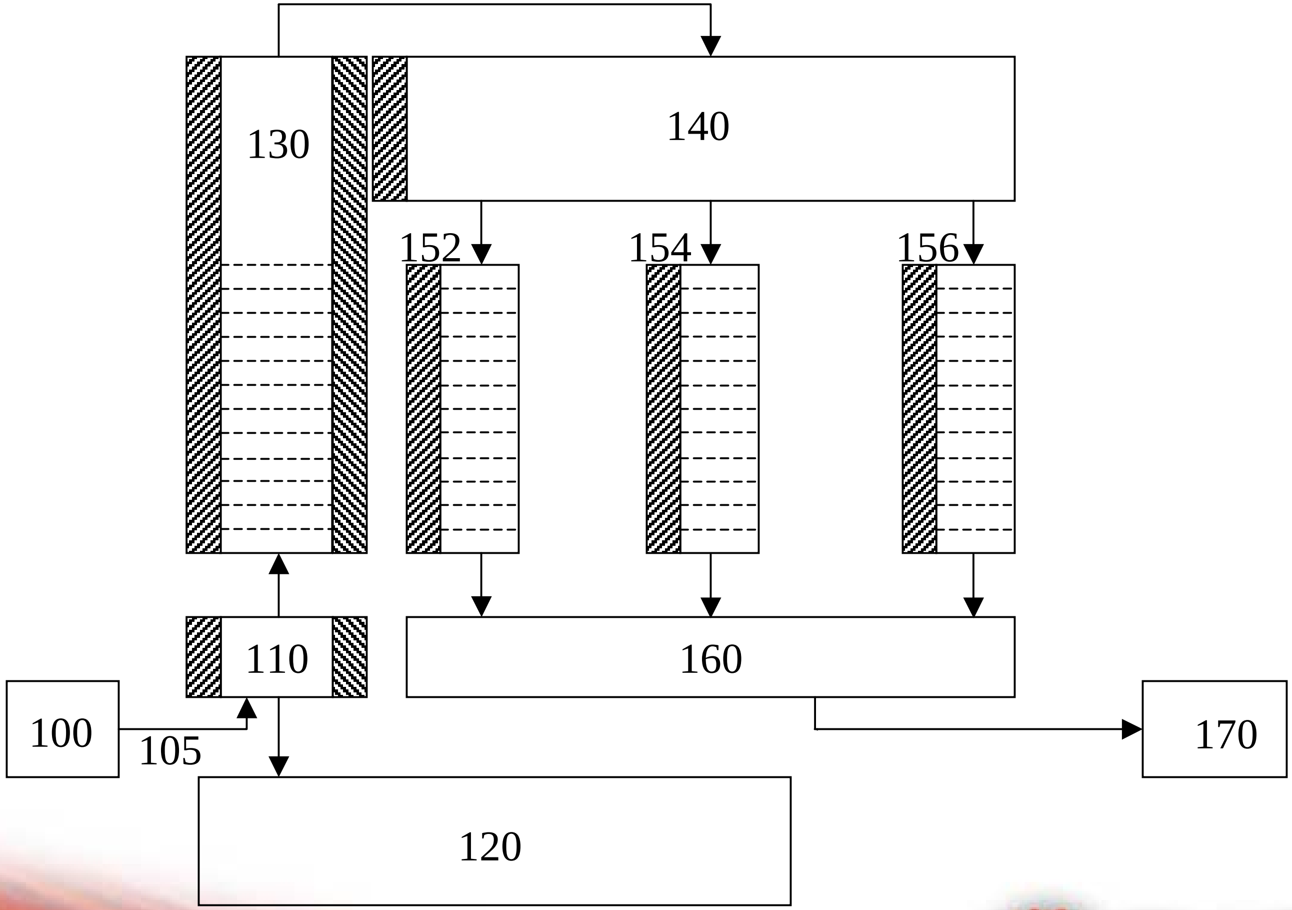
Boundary Conditions…

- Pressure = 7 MPa
- Operating average outlet temperature = 1134 K
- All walls adiabatic
- Inlet velocity = 98.625 m/s (Flow rate = 160 kg/s for ½ plenum)
- Diameter of inlet hole = 200 mm
- Backflow temperature = 1000 K
- Turbulent intensity = 10%

RELAP5 Model Summary: Reactor Vessel Model

- Coolant active and stagnant volumes
- Structures in the core region
  - inner and outer reflectors
  - upper and lower reflectors
  - core barrel
  - upper plenum shield
  - reactor vessel wall and upper head
- Structures below the core are being ignored

VHTR Vessel Hydraulic Nodalization—Bypass Not Shown



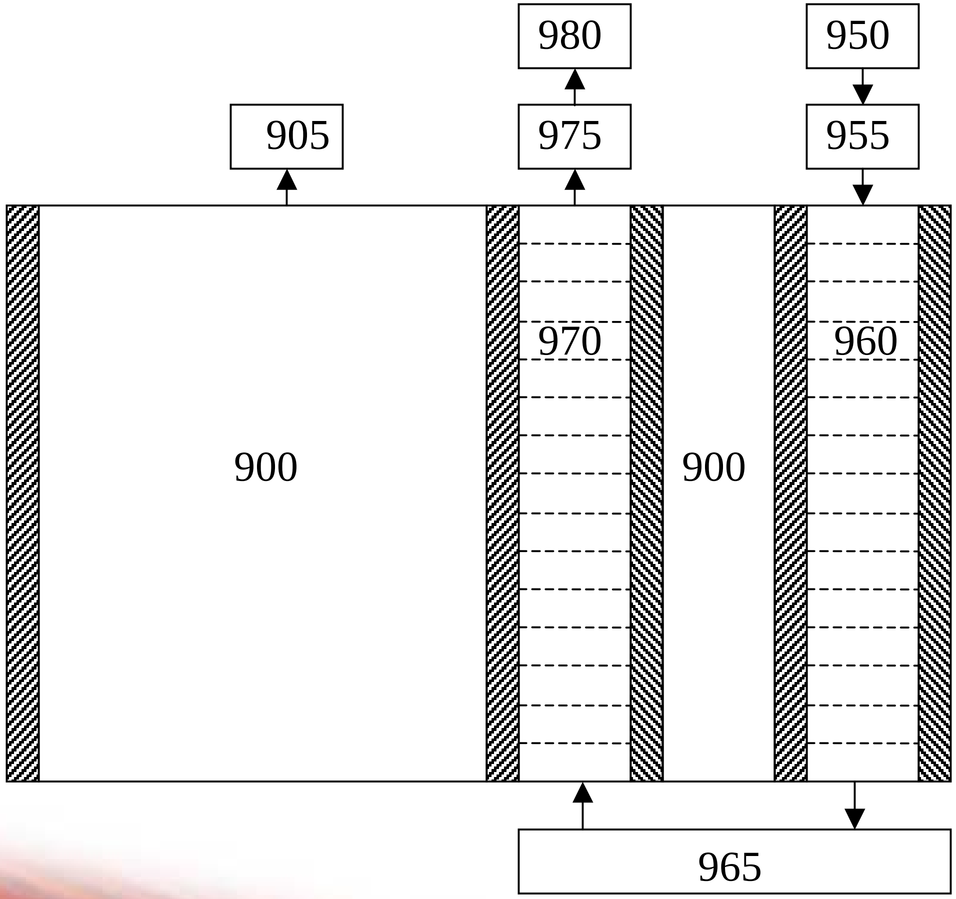
Other Candidate Coupled Calculations

- Reactor cavity cooling system
- Coupled through heat transfer boundary

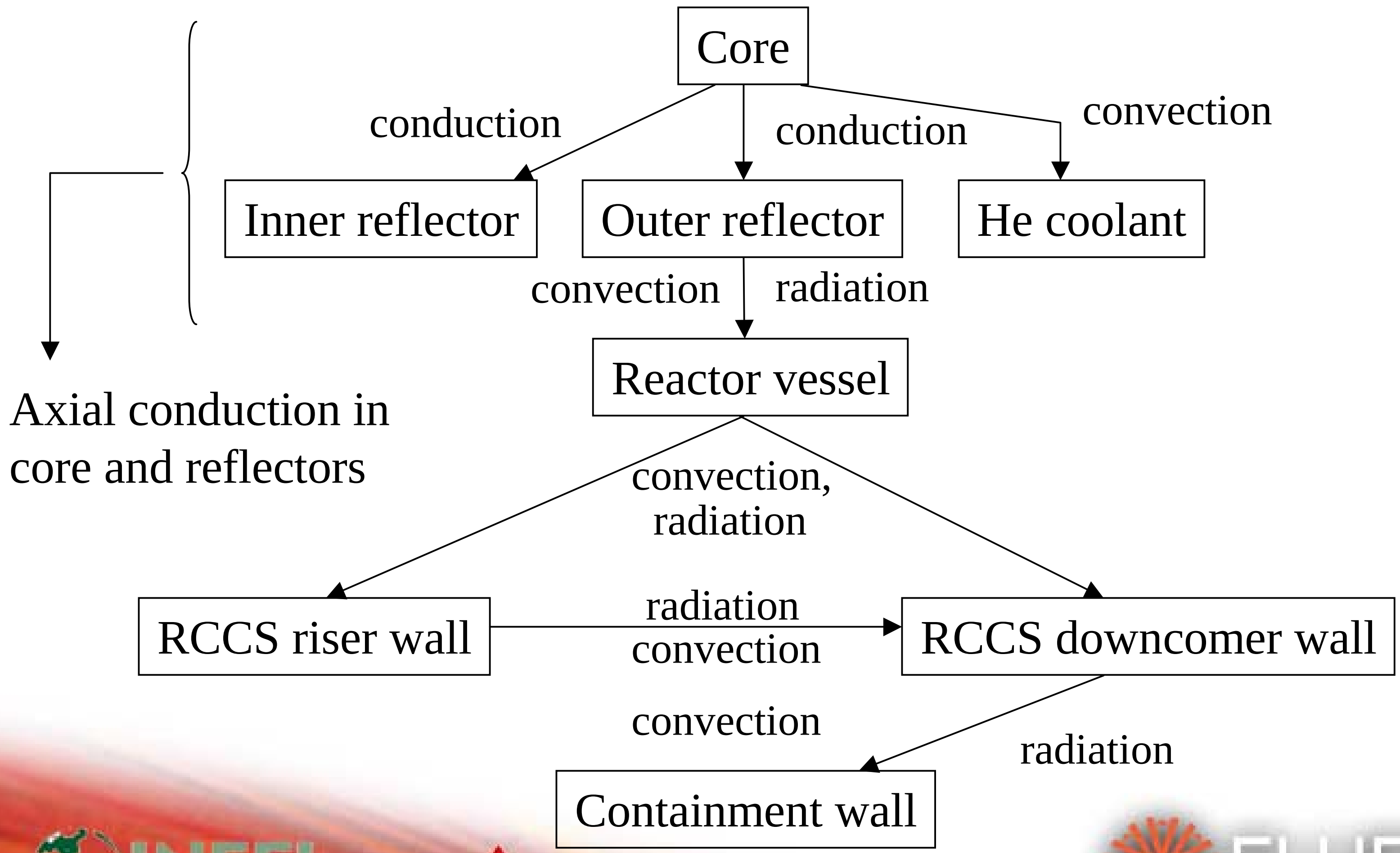
RELAP5 Model Summary: Ex-vessel Model

- Containment air volume
- Reactor cavity cooling system (RCCS)
  - Inlet plenum/downcomer piping
  - Lower distribution plenum
  - Riser/outlet plenum
  - Riser, downcomer, and outer metal walls
- Containment concrete wall and surrounding soil (behind RCCS downcomer)
- Other structures/walls neglected

VHTR Reactor Cavity Nodalization



RELAP5 Model Summary: Heat Transfer Modeling with RCCS Model



Status & Schedule…

- Modifying coupling protocol to work with Fluent 6.1 (work previously done for Fluent 5.5)

- Will perform calculation in Sept on Sun workstation.