Idaho National Engineering and Environmental Laboratory

Calculational Envelope: RELAP5-3D & Fluent CFD Using Segregated Solvers†

Presented by:

Walter L. Weaver

August 25, 2004

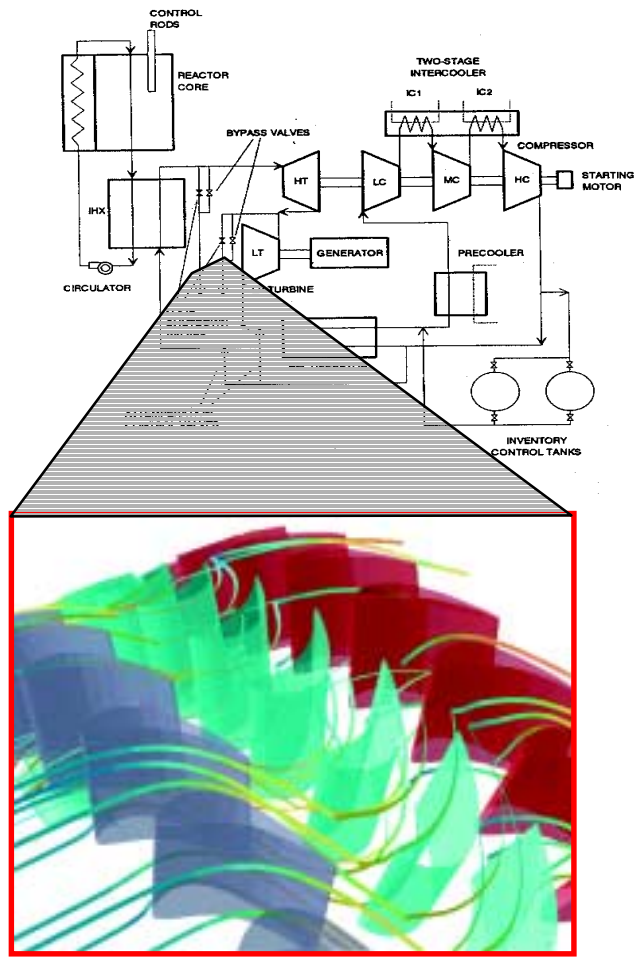
†*Summary of BE2004 paper by Schowalter, et al, "Discussion on Calculational Envelope of Fluent CFD Code & RELAP5 Systems Analysis Code When Using Segregated Solvers"*

Outline

- *Introduction*
  - *Solvers in Fluent*
  - *Background on use of Fluent's solvers*
- *Comparison of Fluent calculation using coupled vs segregated solver.*
- *Summary*

Fluent to RELAP5-3D©/ATHENA Coupling: Advantages

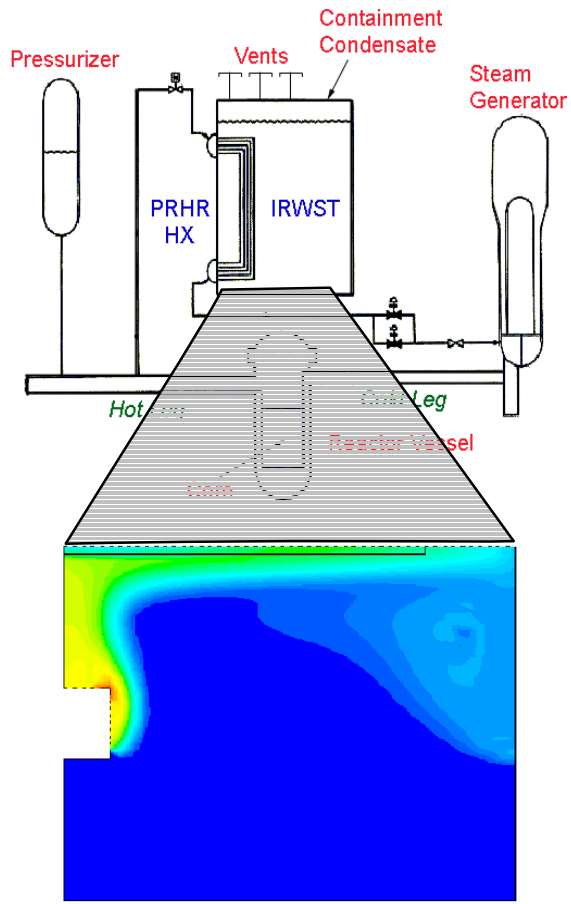
- *Utilize system-wide macro modeling capability of RELAP5-3D/ATHENA*
- *Model some components of the system in detail using the micro-modeling features of FLUENT*



Courtesy of Fluent

Benefits of Fluent & RELAP5-3D©/ATHENA Coupling

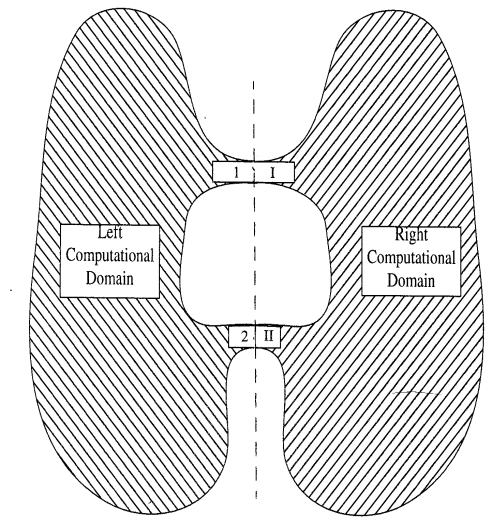
- *The performance of the system depends on the flow through each component, and vice versa*
- *Boundary condition information is transferred back and forth between the two codes*
- *Both the system and component behaviors are more accurately predicted*



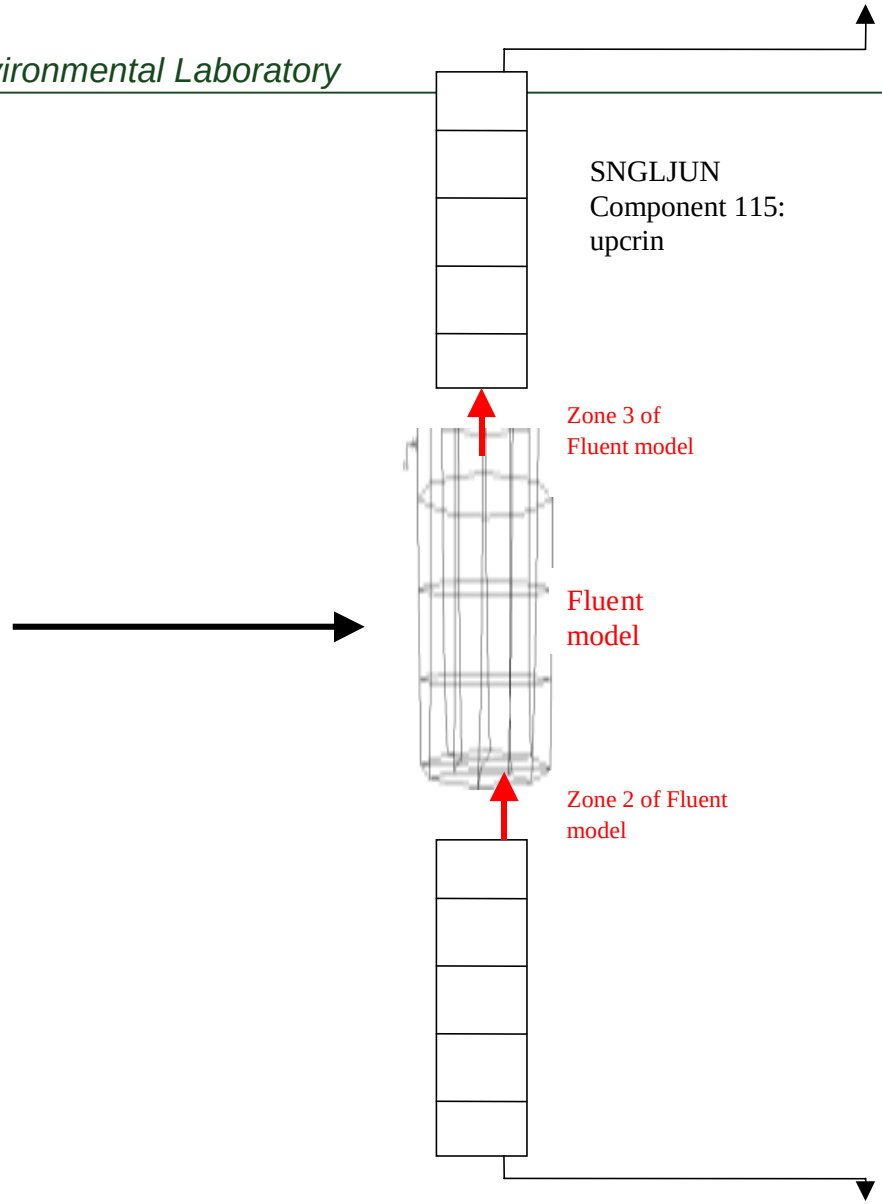
Courtesy of RPI

Coupling of RELAP5-3D©/ATHENA & Fluent is Semi-Implicit

- *RELAP5-3D©/ATHENA & Fluent move forward time step-by-time step to perform calculation.*
- *Calculation only moves forward when convergent solution is achieved in both left and right computational domains.*
- *RELAP5-3D reactor physics can be coupled to Fluent (as one domain—with small additional of work). Completion of this task will create only known example of commercial CFD code with reactor physics.*
- *Any of Fluent's solvers may be used for the coupled calculation.*



Blowup of Fluent model linked to RELAP5-3D© model



But Convergence & Solution Times Dictated by Fluent's Linarization & Solution Methodologies…

- *Fluent uses 3 types:*
  - *Coupled Explicit*
  - *Coupled Implicit*
  - *Segregated*
- *Coupled Explicit & Implicit*
  - *Flow & energy equations are coupled.*
  - *Originally designed to be used for high-speed compressible flows*
  - *Requires 1.5 to 2 times more memory than segregated solver.*
  - *Usually used with very fine meshes.*

Segregated Solver…

- *Originally used for incompressible and mildly compressible flows†.*
- *Default solver in Fluent.*
- *Governing equations solved sequentially:*
  - *Update properties.*
  - *Solve momentum eq*
  - *Solve continuity eq*
  - *Solve energy eq*
  - *Iterate until convergence achieved.*

† *Gas flow considered incompressible with Mach numbers < 0.3.*

In Practice…

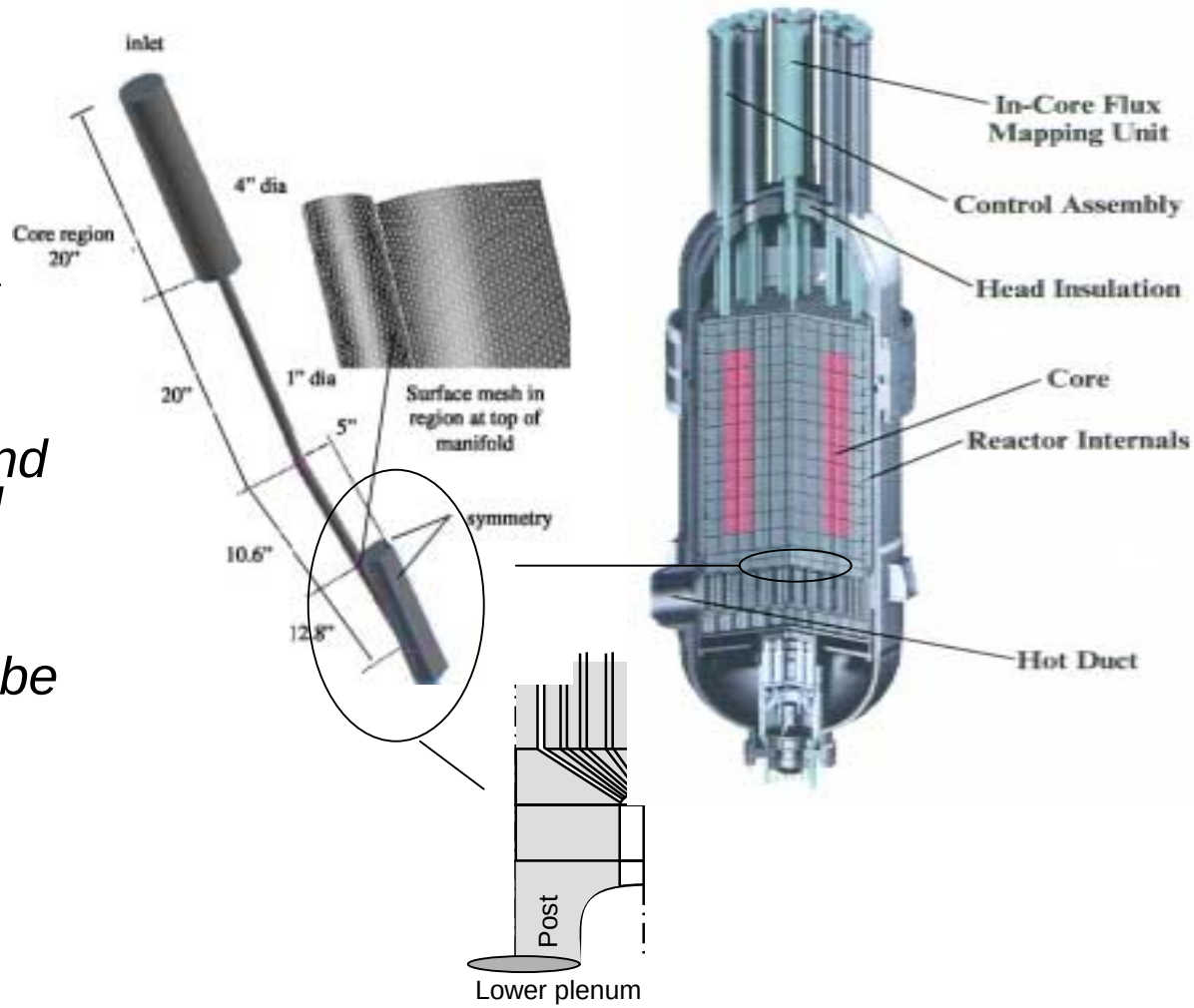
- *All of Fluent's solvers will work with the coupled code.*
- *For our advanced reactor analyses the segregated solver is generally more applicable, e.g.,analysis of mixing in plena at steady-state.*
- *But sometimes it is desirable to analyze transient behavior where large property changes may occur and the analysis may be part of overall calculation.*
- *Since coupled solver usually used for scenarios where large property changes are expected, the question is how do results obtained using segregated solver compare to those obtained using coupled solvers for same scenario.*

Therefore we requested guidance from Fluent on calculational envelope of segregated solvers.

Test Problem Based on General Atomic's GT-MHR Coolant Exit Channel…

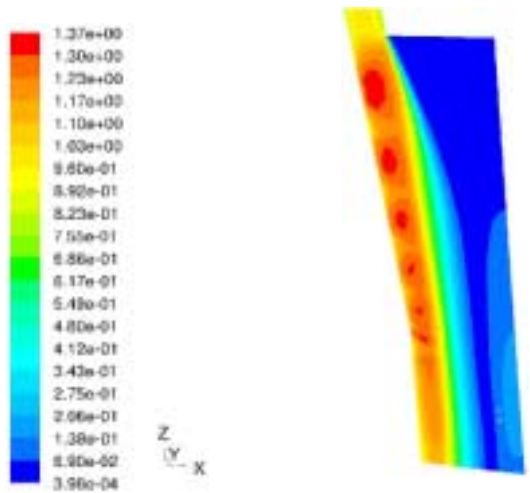
- *Working fluid: helium*
- *Inlet P = 1010 psia &*

  T = 762 °F
- *He equation of state: compressible ideal gas law*
- *Scenario: assumed blowdown underway and flow choked at channel exit.*
- *Investigate whether segregated solver can be used for calculation*
- *Coupled solver results serve as baseline calculation*



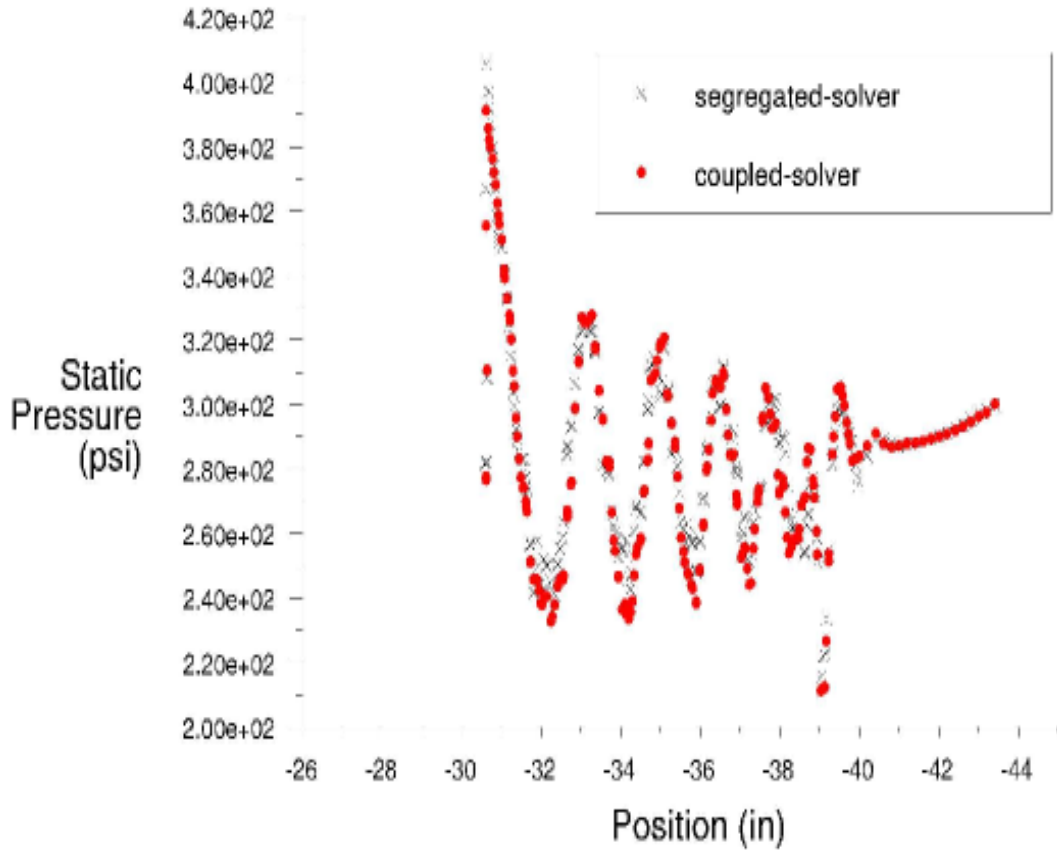
Mach Number Contours Calculated Using Both Coupled & Segregated Solvers …

- *Mesh has 600,000 cells*
- *Plenum volume shown at right.*
- *Mach number contour plots show shock diamonds describing underexpanded jet.*
- *Mach number peaks at ~1.4 in manifold expansion zone.*
- *Results shown at right obtained using segregated solver.*



Comparison of Plenum Pressure Calculated Using Segregated & Coupled Solvers…

- *Both solvers give similar results.*
- *Calculation using coupled solver gives sharper shock structure more typical of expected behavior.*



Fluent's Segregated Solver…

1. *Commonly used for our gas-cooled reactor calculations with assumption helium is incompressible (Mach number less than ~0.3)*
2. *Has a wide range of applicability and can be used to calculate portions of calculation where large changes in properties are expected.*
3. *However, if the most important result sought is for scenario with compressible flows and large changes in fluid properties—coupled solver should be used.*