INL Standard Problem 1: Description & Status

Richard R. Schultz
NGNP Methods Program Manager
Idaho National Laboratory

RELAP5 Seminar
August 16, 2006

Pioneering Science and Technology

Idaho National Laboratory

Overview

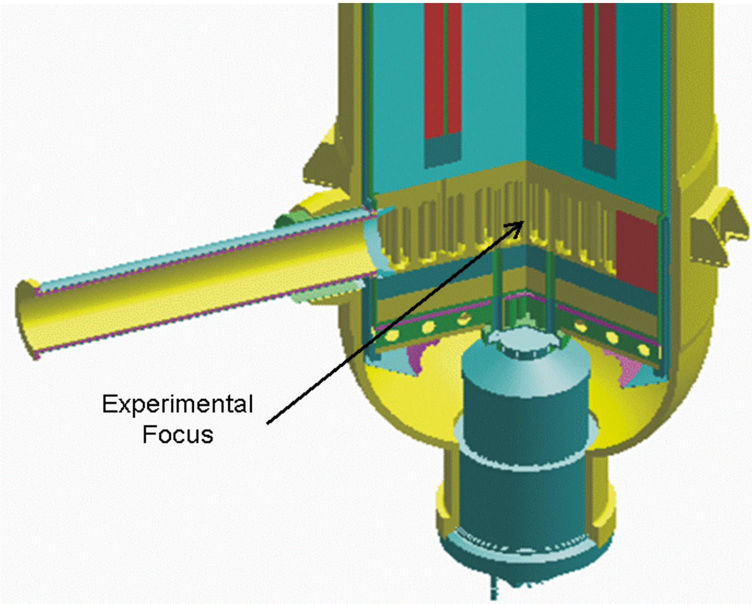
- Objectives: NGNP Methods
- How we are defining needs in NGNP Methods
- Summary description of system
- Normal operation & accident scenarios
- Standard Problem
- Potential issues
- Where do we go from here?

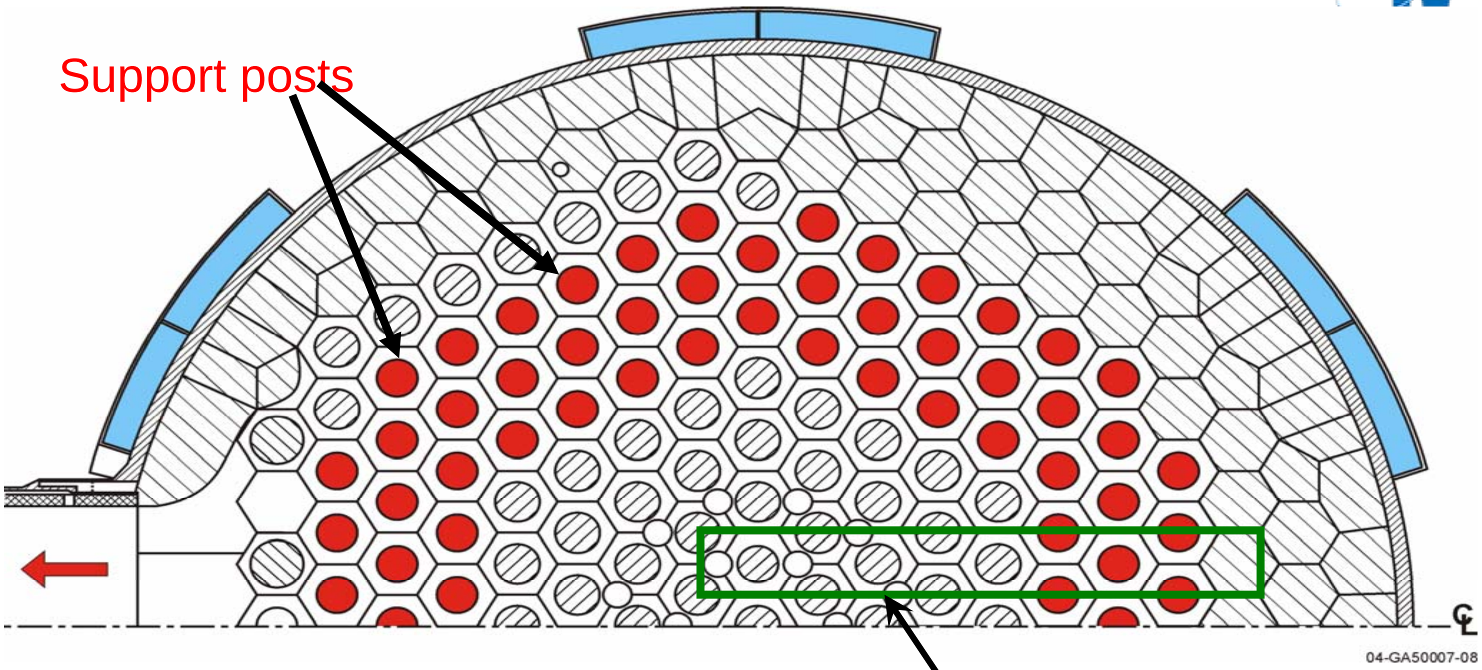
INL Has Been Developing a Candidate Standard Problem

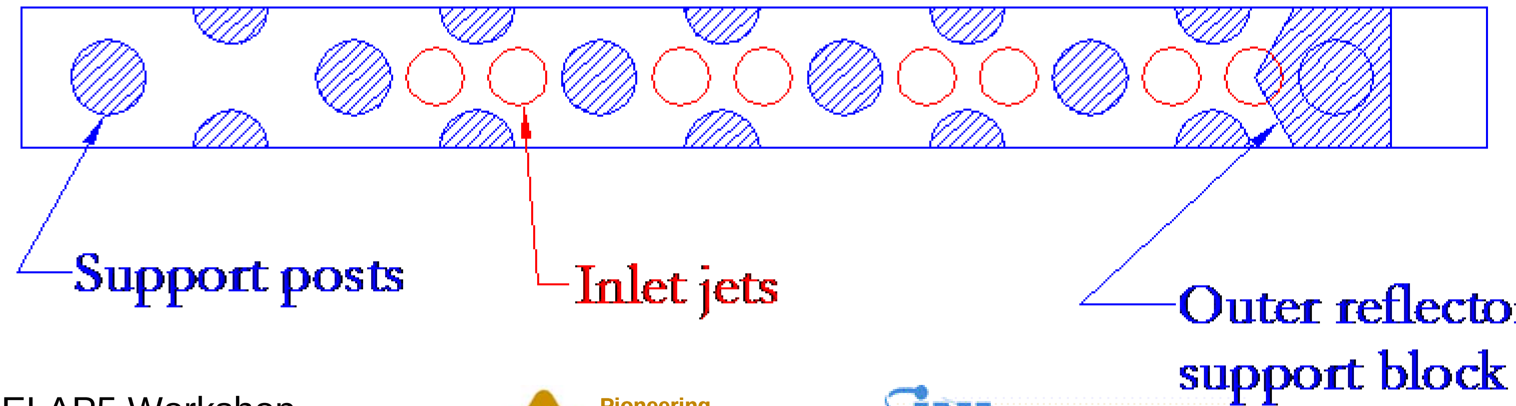
- Based on the need to calculate the flow behavior in the lower plenum for the GT-MHR, experiments are underway that characterize the turbulent mixing and behavior in one specific region of the lower plenum.
- Must develop the initial practices and procedures for performing standard problems using this experiment as the basis. Development of practices and procedures will be an iterative process.
- Therefore, wish to implement practices and procedures and evaluate whether changes in methodology are required.
- Practices and procedures include (a) requirements for performing standard problem experiments and (b) requirements for performing CFD analyses
- Exercise standard problem methodology—improve methodology based on findings for first standard problem
- Similar methodology will be used for RELAP5.

Experiment Has Been Defined Using Rigorous Methodology

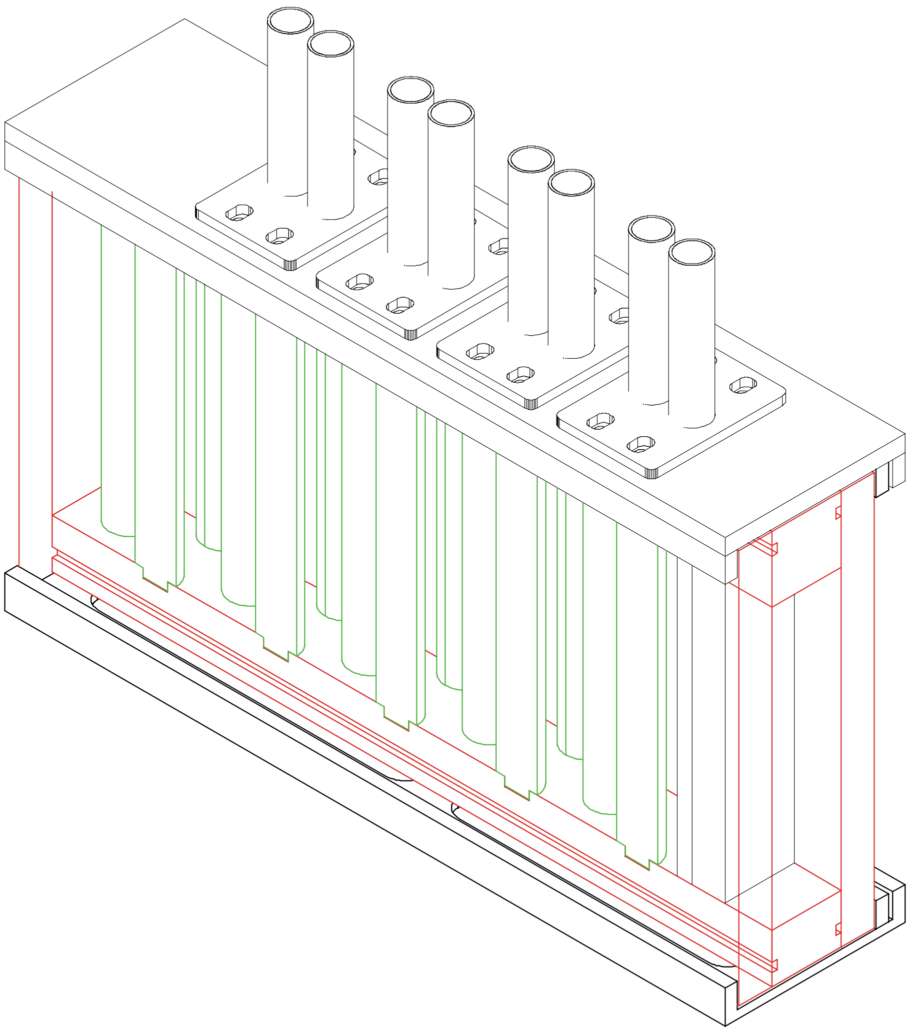
- Scaling analysis performed to link scaled experiment to plant lower plenum.
  - Appropriate range of jet velocities (V_{Jet}/V_{Plenum})
  - Turbulent flow in jet inlet channels (L/D ~ 4 & Re_{Jet} > 3500
  - Turbulent or mixed turbulent flow in lower plenum
  - Based on desired height-to-diameter ratio of about 7
  - Resulting model
    - D_P = 1.25 in (31.8 mm)
    - $p/D_P = 1.7$
    - $H_{Plenum}/D_P = 6.85$
    - $D_{Jet}/D_P = 0.7$
  - For design details see McEligot, et al., 2005, "Development of an Experiment for Measuring Flow Phenomena Occurring in a Lower Plenum for VHTR CFD Assessment," INL/EXT-05-00603.
- Rigorous experimental uncertainty analysis underway.




Support posts
Present Model
04-GA50007-08


Support posts
Inlet jets
Outer reflector support block

Apparatus Drawing

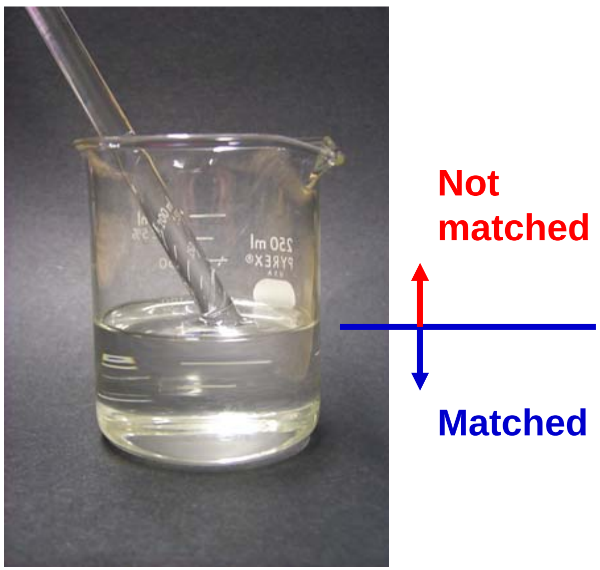


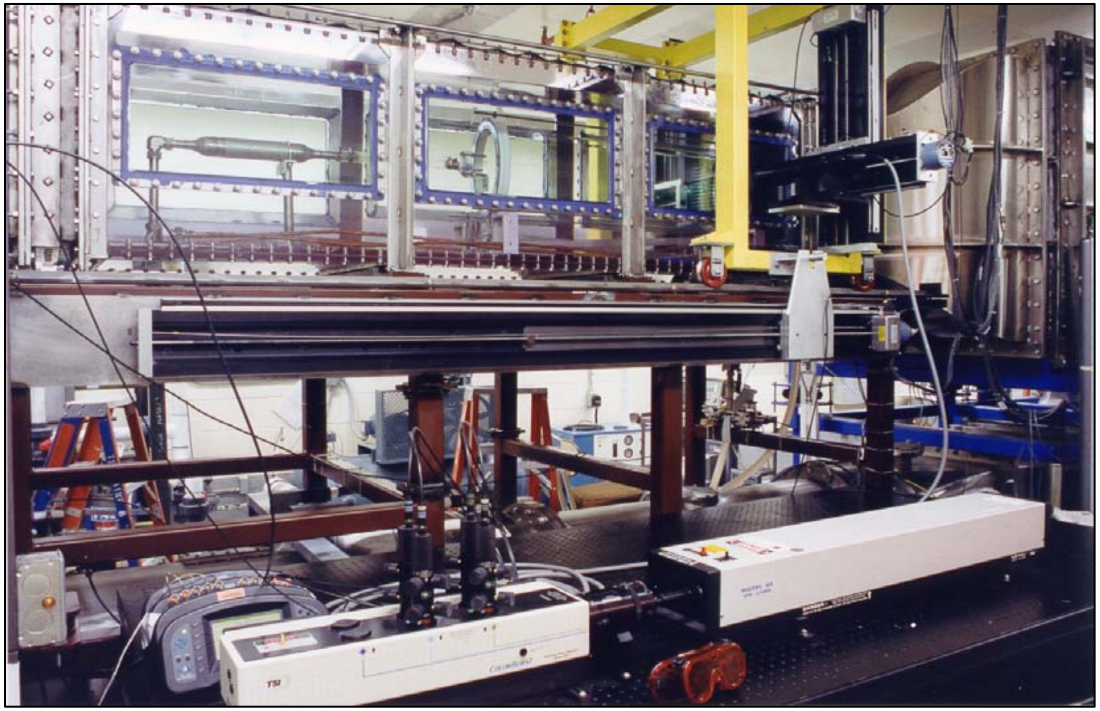
Picture of 1st Standard Problem Apparatus

INL's MIR Experimental Facility

- **Matched index of refraction (MIR) and optical measuring techniques allow flow measurements for complex flow geometries**

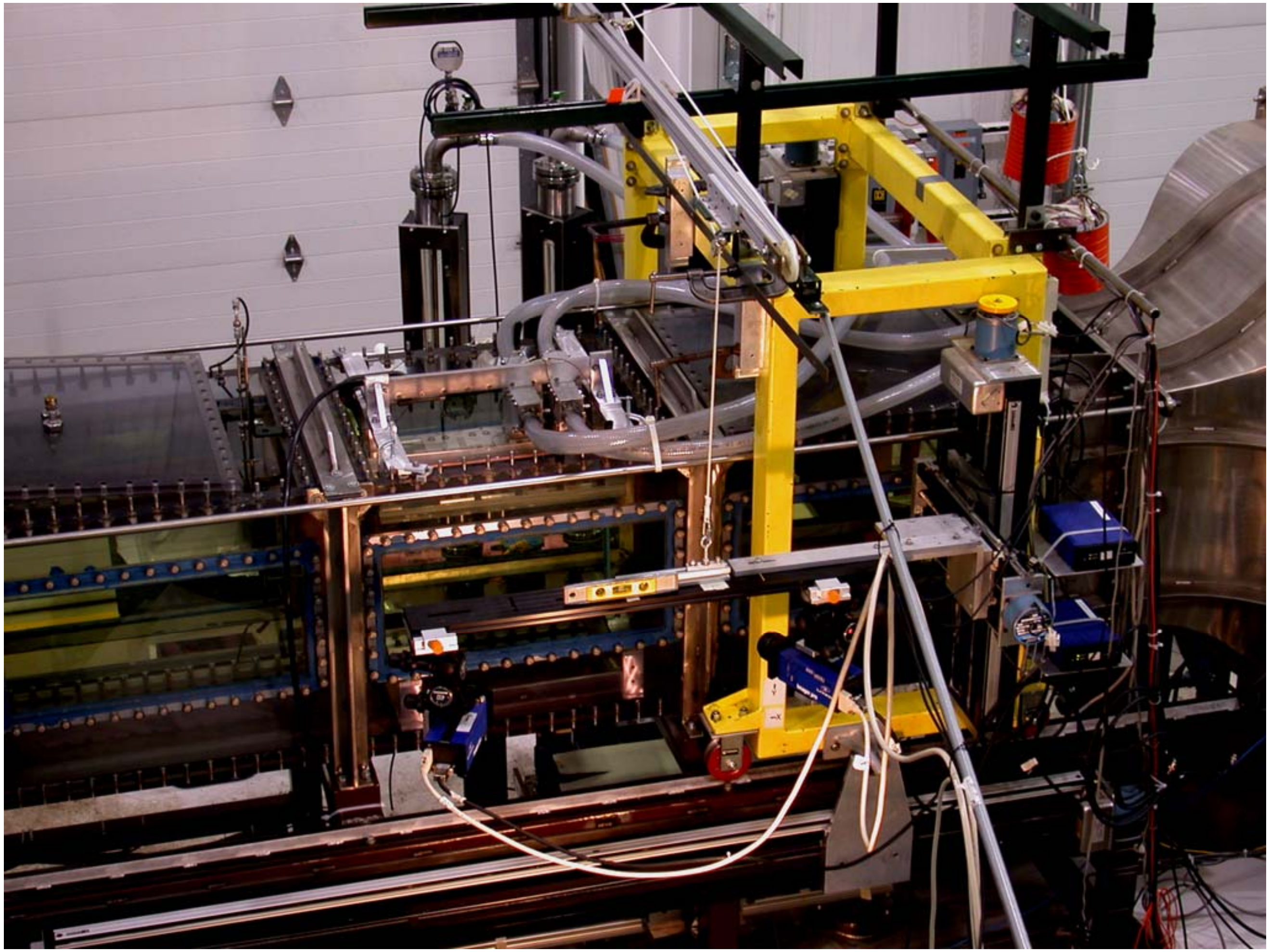
Example of application of refractive-index-matching



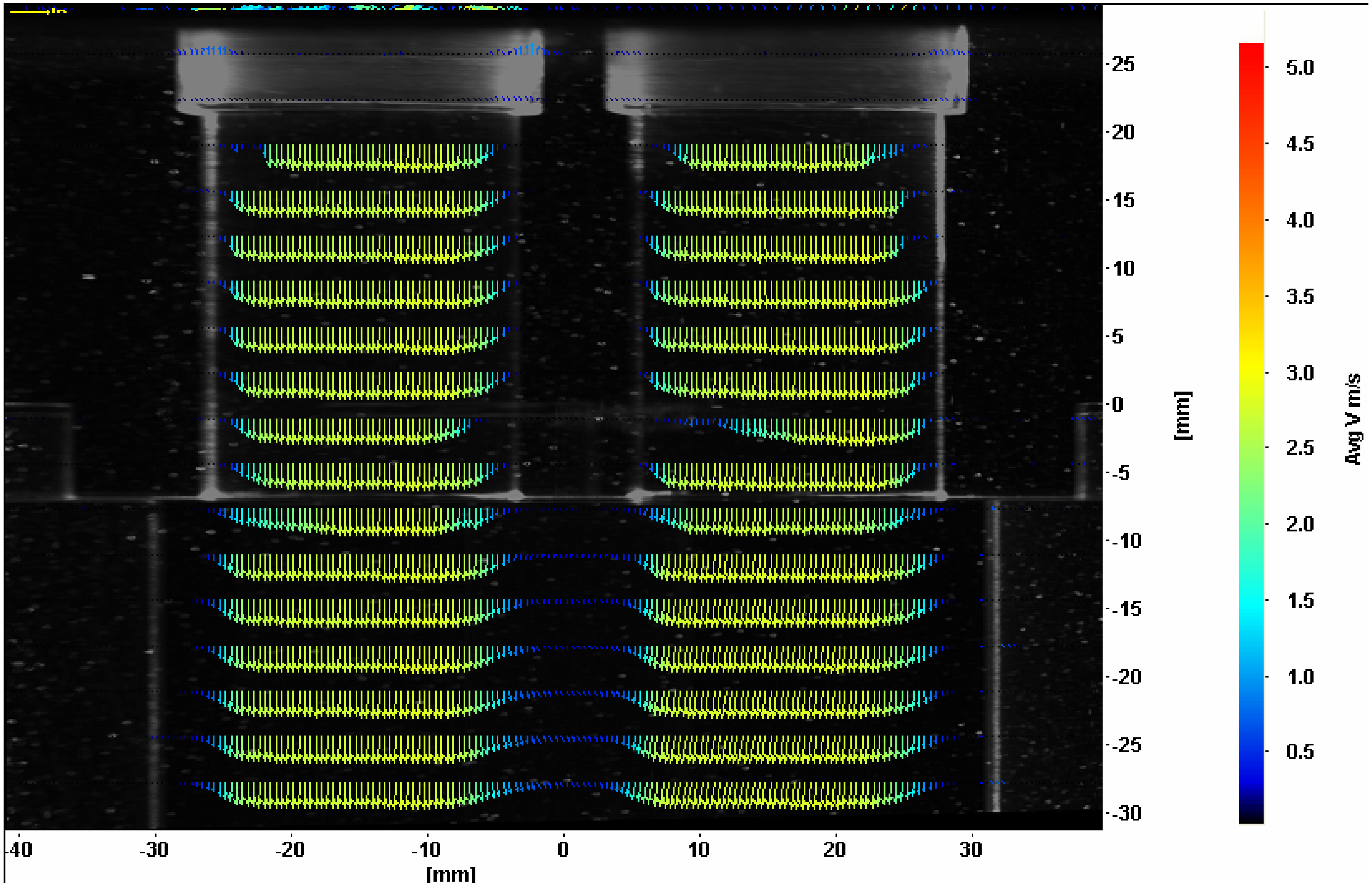


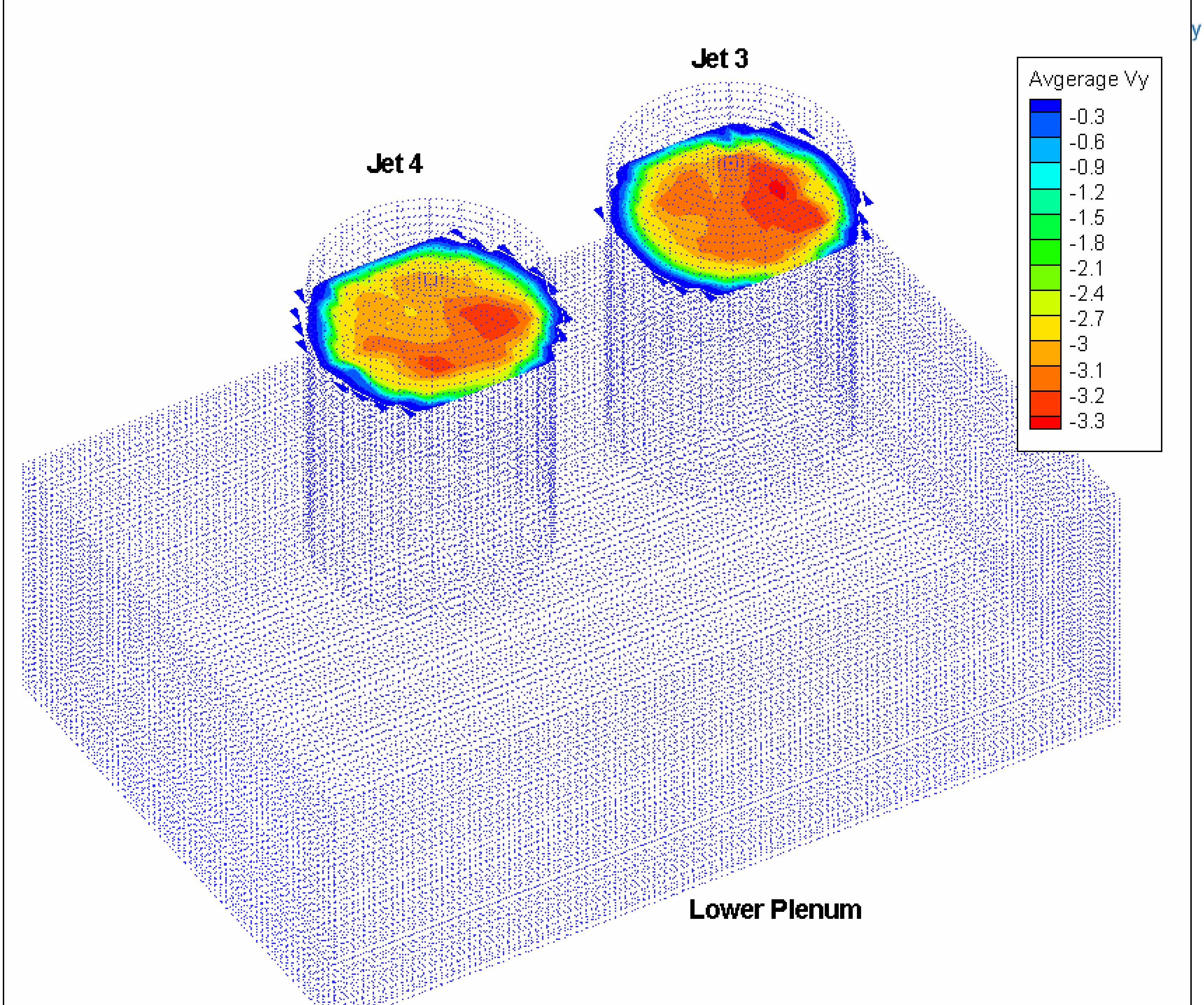
INL's MIR facility (world's largest)

Lower Plenum Model Installed in Test Section

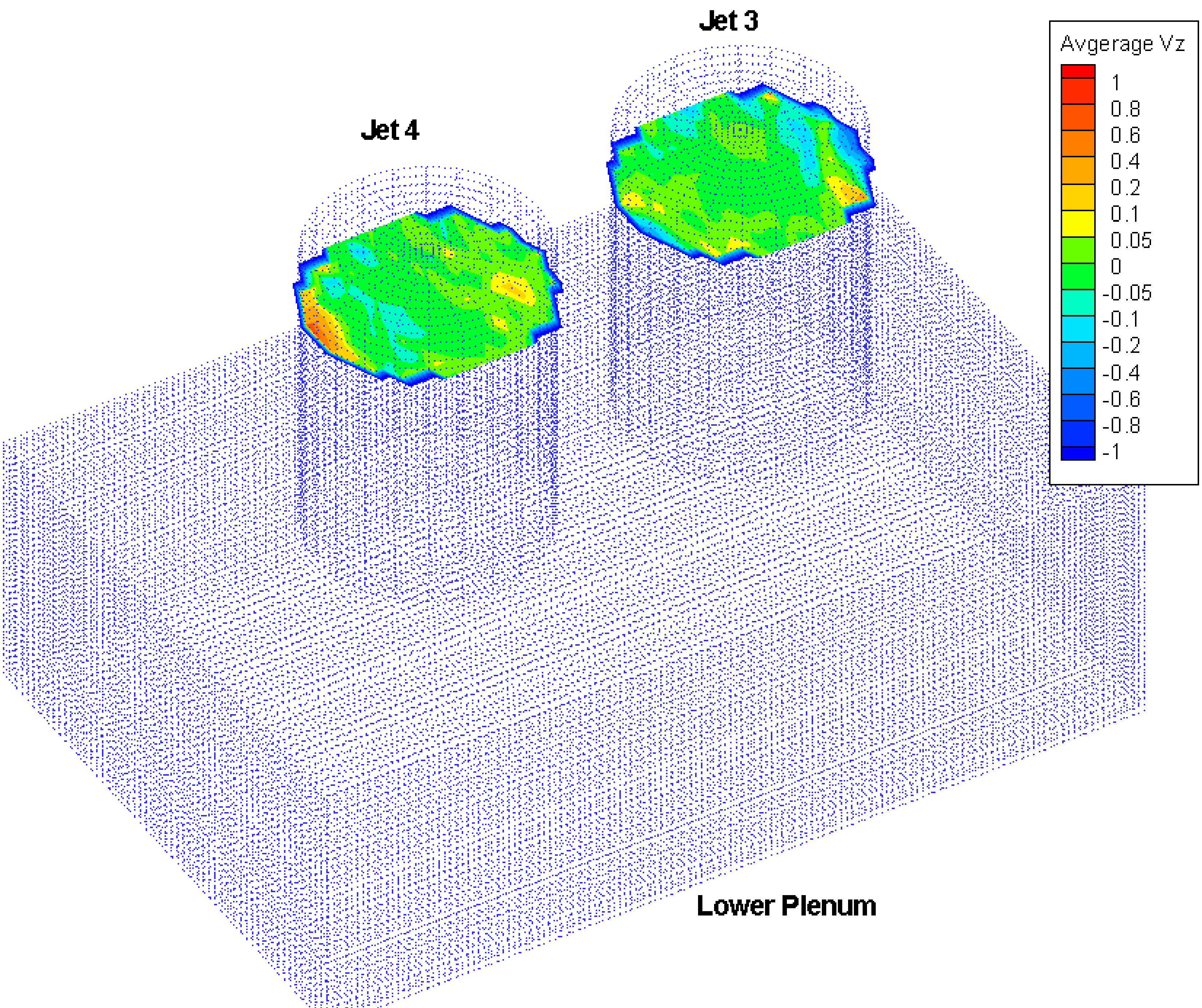


Modified Vector Display

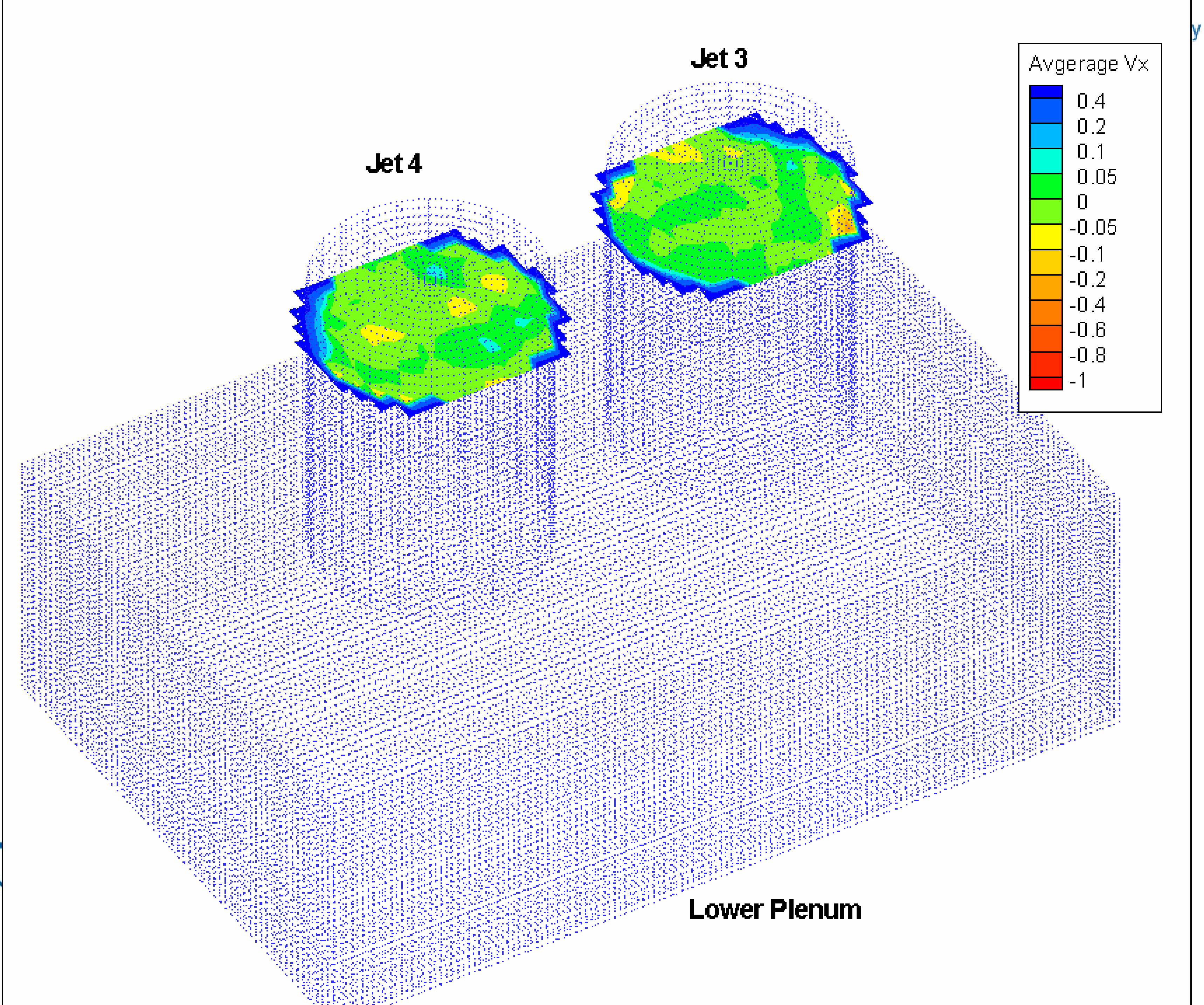




Jet 3
Jet 4
Avgerage Vy
-0.3
-0.6
-0.9
-1.2
-1.5
-1.8
-2.1
-2.4
-2.7
-3
-3.1
-3.2
-3.3
Lower Plenum

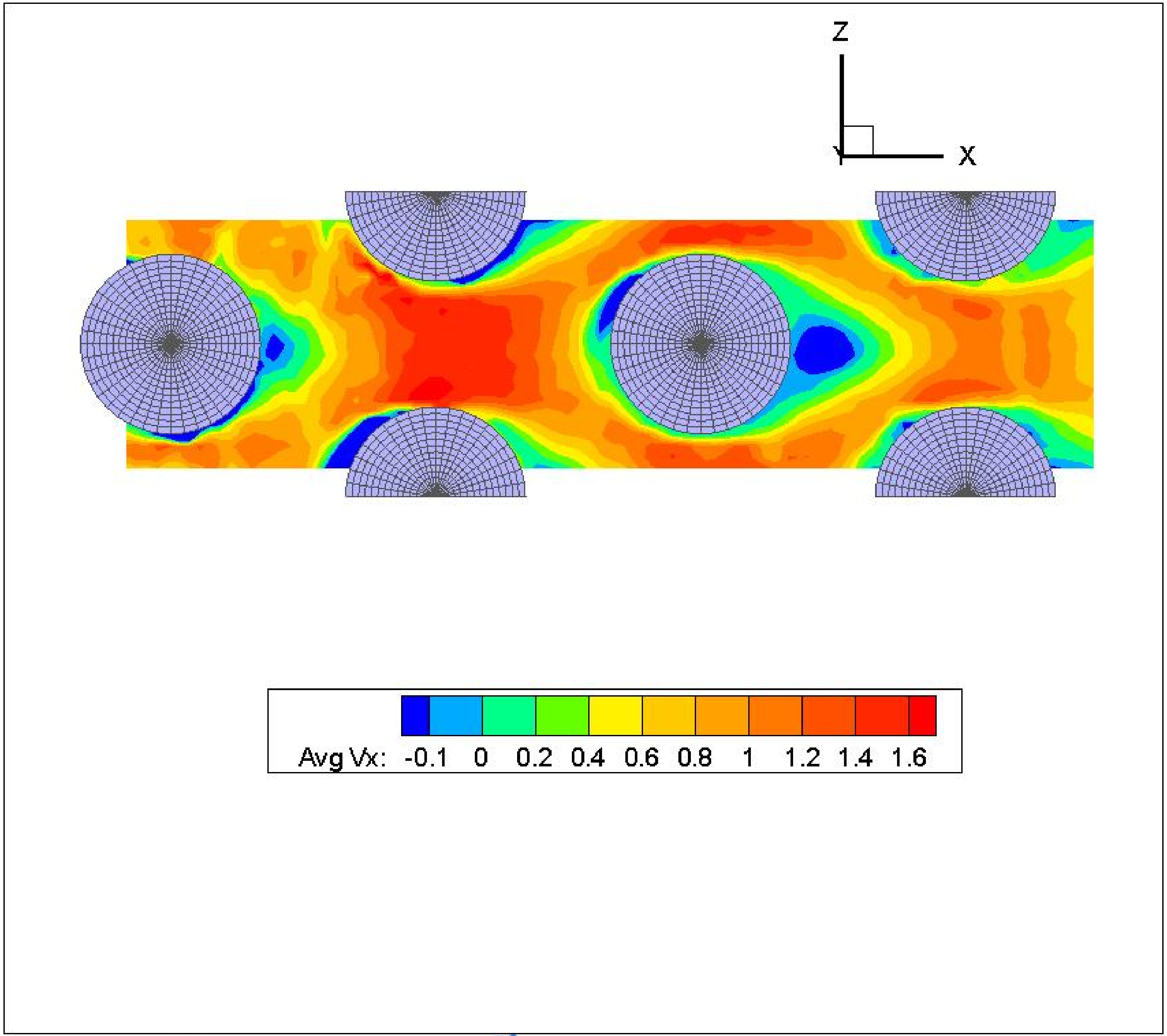


Jet 3
Jet 4
Avgerage Vz
1
0.8
0.6
0.4
0.2
0.1
0.05
0
-0.05
-0.1
-0.2
-0.4
-0.6
-0.8
-1
Lower Plenum



Jet 3
Jet 4
Avgerage Vx
0.4
0.2
0.1
0.05
0
-0.05
-0.1
-0.2
-0.4
-0.6
-0.8
-1
Lower Plenum

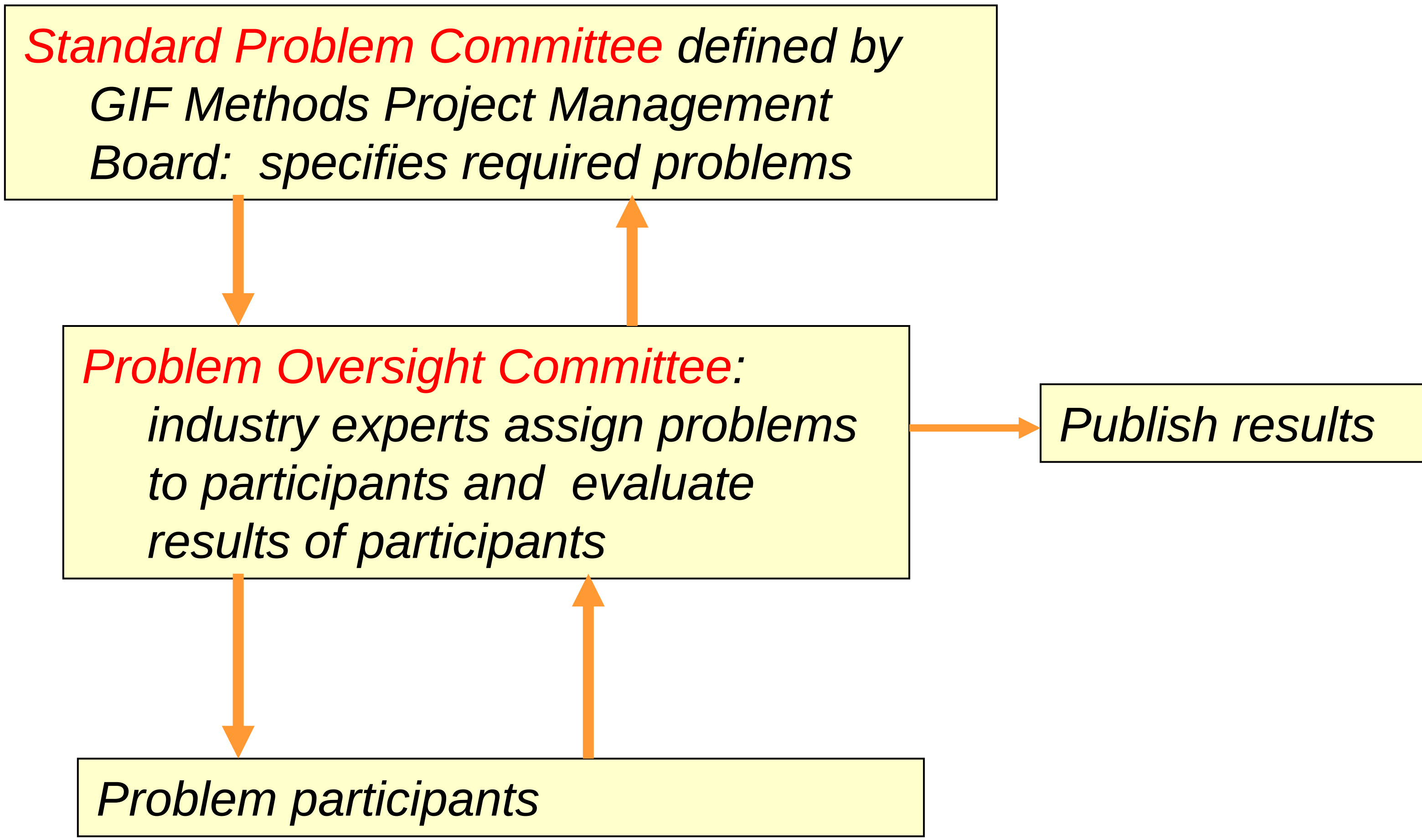
23 planes @ 2 mm intervals



Generation IV Project Management Board Approved INL Standard Problem 1…

- INL will provide relevant data and related material to GIF and also US Problem Oversight Committee for distribution.

Capabilities of CFD & Systems Analysis Codes Validated Using Standard Problems…



An Oversight Committee is in place…

- Industry & Academia:
  - Professor Ismail Celik, West Virginia University; CFD numerical uncertainty, turbulent mixing, and CFD development
  - Professor Yassin Hassan, Texas A&M University; specializes in CFD validation experiments and CFD development
  - Dr. Patrick Roache, Hermosa; specializes in CFD V&V and development
- INL/ANL:
  - Dr. Richard Johnson, INL; Committee Chair, specializes in CFD development and validation
  - Dr. David Pointer, ANL; specializes in CFD development.
  - Richard Schultz, INL; specializes in standard problem definition and validation.
- US Nuclear Regulatory Commission:
  - Invited to participate

Practice & Procedures for CFD Analysis

1. CFD codes used in licensing nuclear reactors should be verified before being used for design or analysis. Our baseline assumption, for performance of Standard Problem 1 is that CFD codes are verified.
2. CFD code and calculations should be documented.
3. CFD simulations should be performed in such that numerical error is minimized in order to obtain results that are sufficiently accurate to be useful.
4. Uncertainty should be quantified to give a confidence level to the calculations.
5. Experimental data should be used to validate numerical calculations.

Code Verification

- Code verification is a mathematical exercise. The question to be answered is: Are the equations being solved correctly?
- The Method of Manufactured Solutions* can be used to compare exact errors on a series of refined grids. (You make up a solution and add a source term to the equations such that the solution is the solution to the modified equations.) This verifies:
  - any equation transformations (e.g. boundary-fitted coordinates)
  - the (observed) order of the discretization
  - the coding of the discrete equations
  - the matrix solving procedure
- The code verification is something that is performed once (unless the code is modified)

* Roache, Patrick J., Verification and Validation in Computational Science and Engineering, Hermosa, Albuquerque, 1998.

Code & Calculation Documentation

The following should be documented:

- The equations being solved
- Coordinate transformations
- The discretization method
- Model equations (turbulence, etc.)
- The solver
- Other numerical techniques used
- Boundary and Initial Conditions

Practices & Procedures to Minimize Numerical Error

The following are required by ASME Journal of Fluids Eng.:

- Methods must be at least second order accurate in space.
- Inherent or artificial viscosity (or diffusivity) must be assessed and minimized.
- Grid independence or convergence must be established.
- When appropriate, iterative convergence must be addressed.
- In transient calculations, phase error must be assessed and minimized.

Calculation Verification

Calculation verification is seen by some to be the same as quantification of numerical uncertainty. Some additional information must be obtained to estimate uncertainty.*

The most straightforward way to quantify numerical uncertainty is to obtain calculations on 2, 3 or 4 different grids and then use Richardson extrapolation to obtain an estimate of the numerical error. Then an uncertainty band can be calculated such that 95% of values calculated are within the uncertainty band. The JFE provides a procedure for doing this. See Statement of Numerical Accuracy on the JFE Website.

* Roache, Patrick J., Verification and Validation in Computational Science and Engineering, Hermosa, Albuquerque, 1998.

Calculation Validation

Compare results of calculations to experimental data that is similar to the flow in question, has a reasonable experimental uncertainty and is fully documented to be useful for CFD calculation validation, including boundary and initial conditions being completely specified.

Analysis Practices Should be Well-Defined

For CFD software:

- Oversight Committee will define policy; probably extension of that described in Journal of Fluids Engineering (Vol 115, 1993)

- Benchmark study requirements will be extensions of those described in: C. J. Freitas, "Perspective: Selected Benchmarks from Commercial CFD Codes," Journal of Fluid Mechanics, 117, 1995, pp. 208 to 218.

For systems analysis software—use directly practices and procedures implemented by USNRC.

Schedule: Standard Problem 1

- Release of Standard Problem 1 initial and boundary conditions: September 30, 2006 to November 30, 2006. Required practices and procedures will be released concurrently.
- Potential participants: unrestricted—however, bona fide CFD software must be used.
- Completion of Standard Problem 1 by participants: within 6 months of problem release.
- Evaluation of results obtained from participants: within 3 months.
- Completion of exercise: September 30, 2007

Potential Issues

- Are the proposed practices and procedures for conducting standard problem experiments and for conducting CFD analyses too restrictive?
- Is the size of the full-sized system (~7 m diameter vessel) too large to enable rigorous CFD analysis? Thus can only qualitative CFD analyses be performed?
- Regarding the numerical accuracy of CFD analyses for the NGNP: what is “good-enough”?
- What are the requirements that must be enforced such that: several users analyzing the same problem using the same CFD software obtain the same answer (within an acceptable bound)?