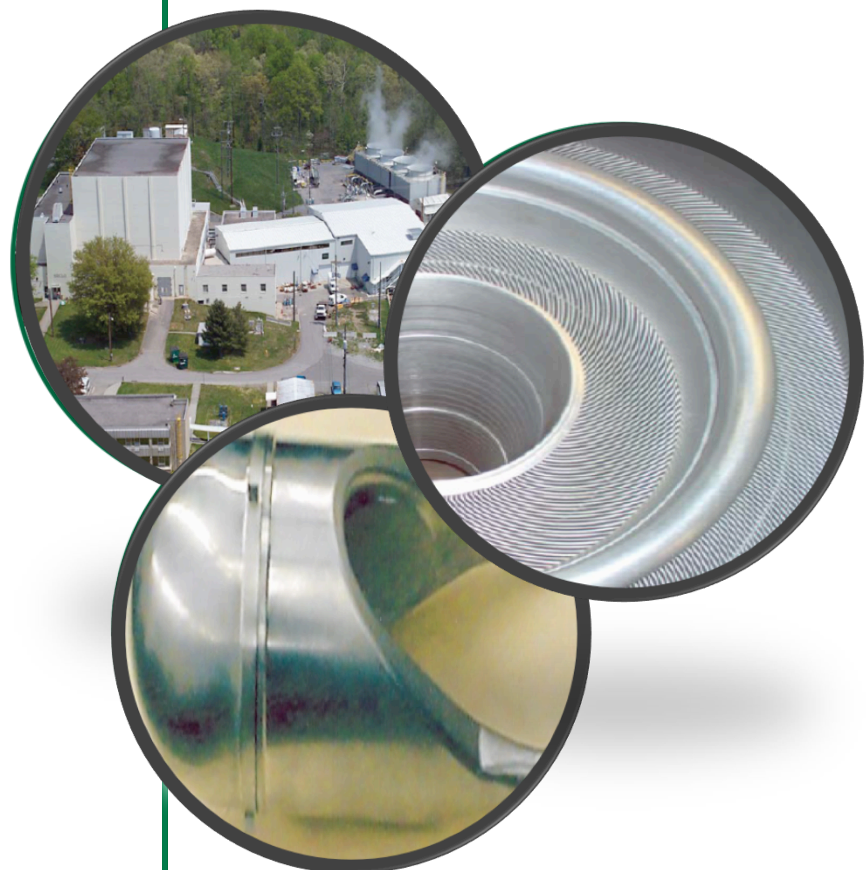
Multi-physics Modeling to Support HFIR LEU Fuel Conversion

presented by:

James D. Freels

Oak Ridge National Laboratory
Nuclear Science and Engineering Directorate
Research Reactor Division
High Flux Isotope Reactor
Nuclear Safety Group

freelsjd@ornl.gov
865-576-8645



presented to:

Joint Meeting of
TRTR/IGORR Conference
Knoxville, Tennessee, USA

September 20, 2010

U.S. DEPARTMENT OF ENERGY

HFIR

OAK RIDGE NATIONAL LABORATORY
MANAGED BY UT-BATTELLE FOR THE DEPARTMENT OF ENERGY

Topics today include:

- multi-physics and why we have chosen COMSOL,
- past success of COMSOL with HFIR,
- multi-physics research goals for HFIR LEU conversion,
- current status, and
- some results

COMSOL is our choice because: part 1 of 2

- Finite-element methods are (arguably from advocates of other methods) the most accurate numerical simulation tool available for deterministic solutions.

- COMSOL is a 100%-true finite-element method code for all the physics to be simulated by COMSOL.

- COMSOL is (arguably) the only true-multiphysics code commercially available (weakly-compressible NS, heat transfer, structural mechanics for this project).

COMSOL is our choice because: part 2 of 2

- If desired, the standard equations solved may be altered on *INPUT* by the *USER* (for example, constituitive equations including turbulence model).

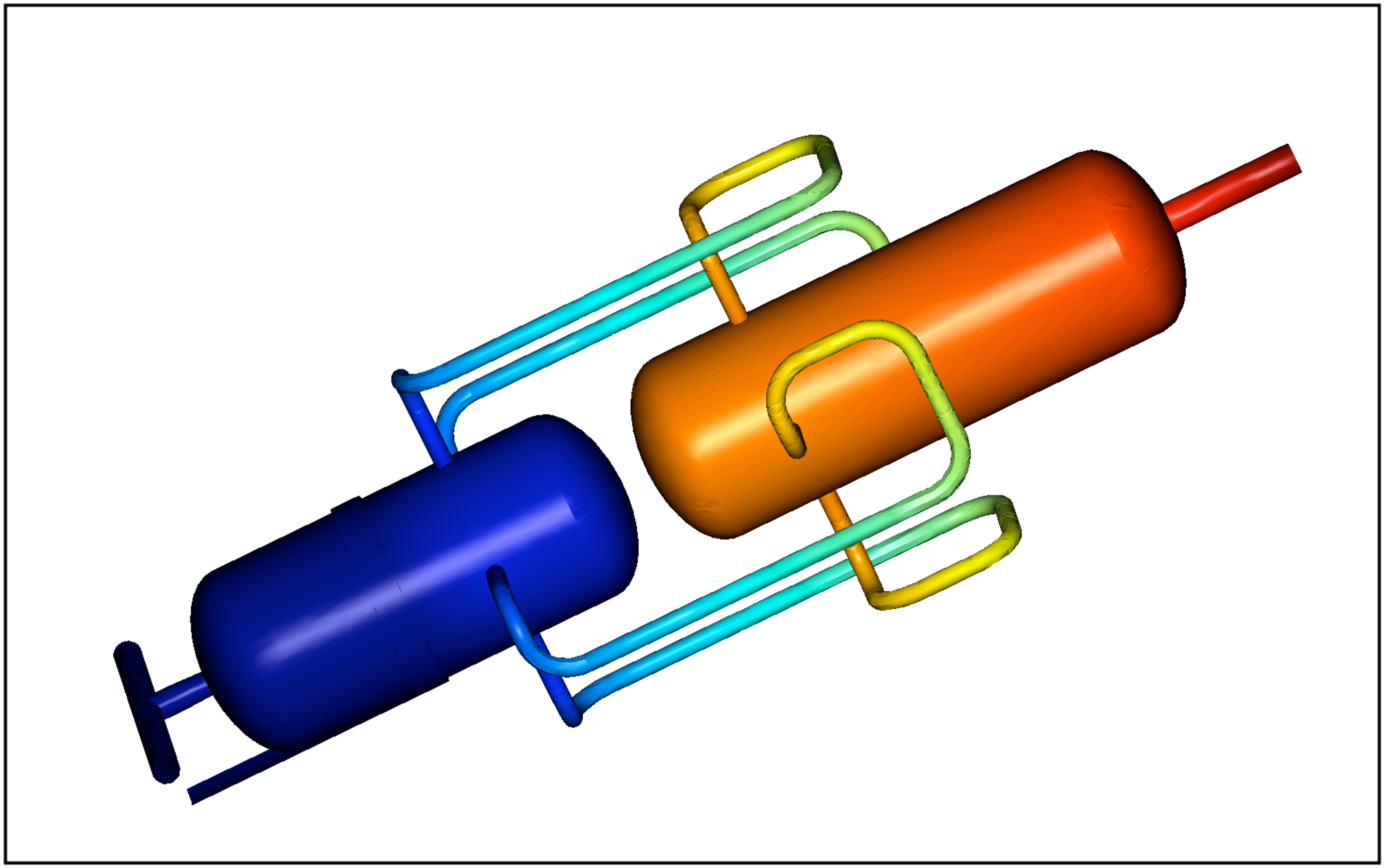
- COMSOL provides a convenient GUI in modern computing environments.

- COMSOL continues to provide technical support and code improvements at a remarkable pace (distributed parallel processing, new modules, interface tools, response to user requests, etc.)

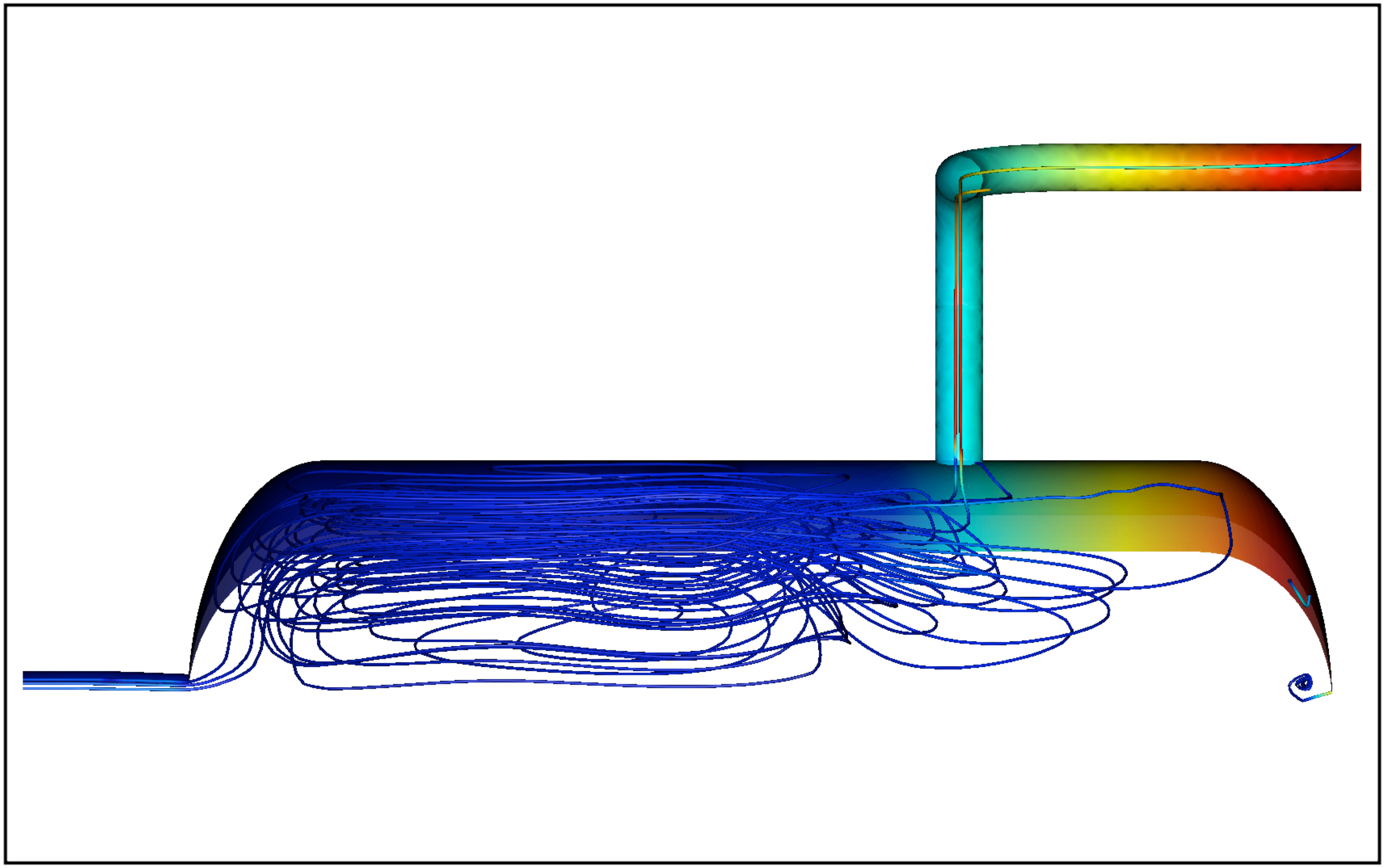
Success Story: COMSOL (FEMLAB)on the HFIR Cold Source Project

COMSOL was the design basis tool for the cold source pressurizer

Success Story: COMSOL multiphysics simulation of the HFIR cold source pressurizer Part 1 of 2



Success Story: COMSOL multiphysics simulation of the HFIR cold source pressurizer Part 2 of 2



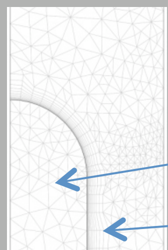
Fundamental goals for COMSOL on the HFIR LEU Conversion Project

- Provide an accurate (best-estimate) 3D simulation of:
  - the fuel-plate and coolant heat transfer,
  - the structural response due to thermal expansion (TSI),
  - the structural response due to fluid interaction (FSI),
  - other physics as necessary (corrosion, radiation effects, etc.).
- Provide perturbed-estimates to the best-estimate models to answer safety-analysis questions (hot spots, buckling potential, reduced flow, decay heat, etc.).

COMSOL Model Development Approach on the HFIR LEU Conversion Project

- **Develop physics test bed in 2D to feed results to the 3D models.**
- **Develop 3D geometry, mesh, and property inputs.**
- **Perform separate-effects simulations in 3D to V&V physics goals (heat transfer, FSI, TSI).**
- **Provide perturbed-estimates to the best-estimate models to answer safety-analysis questions (hot spots, buckling potential, reduced flow, decay heat, etc.).**
- **Provide input data for other parts of the HFIR SAR analysis (transients using RELAP5, etc.).**

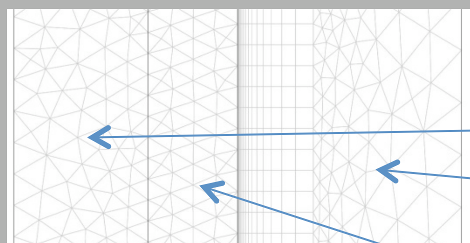
Use a 2D Model as a Test Bed for 3D -start with meshing (triangles)



entrance region is rounded

0.050”/2 symmetric clad width

0.050”/2 coolant channel width

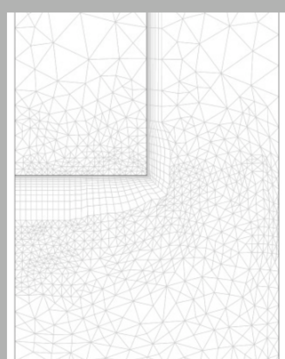


typical main channel section.

fuel (0.030”/2 symmetry)

coolant with boundary layer mesh adjacent to the clad wall

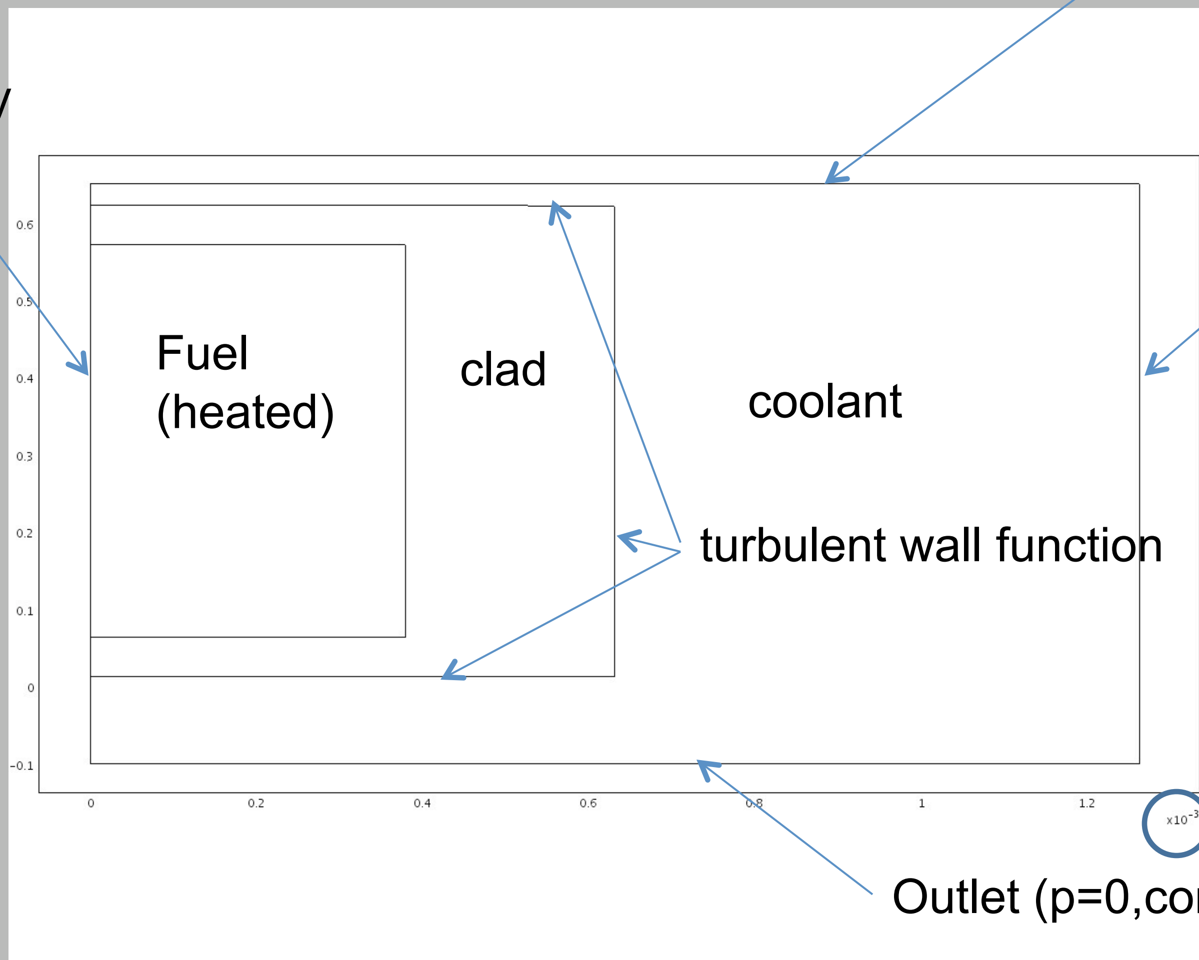
clad (0.010” thick)



exit region is not rounded
extra mesh density in both the entrance and exit regions to capture the large changes in fluid velocity

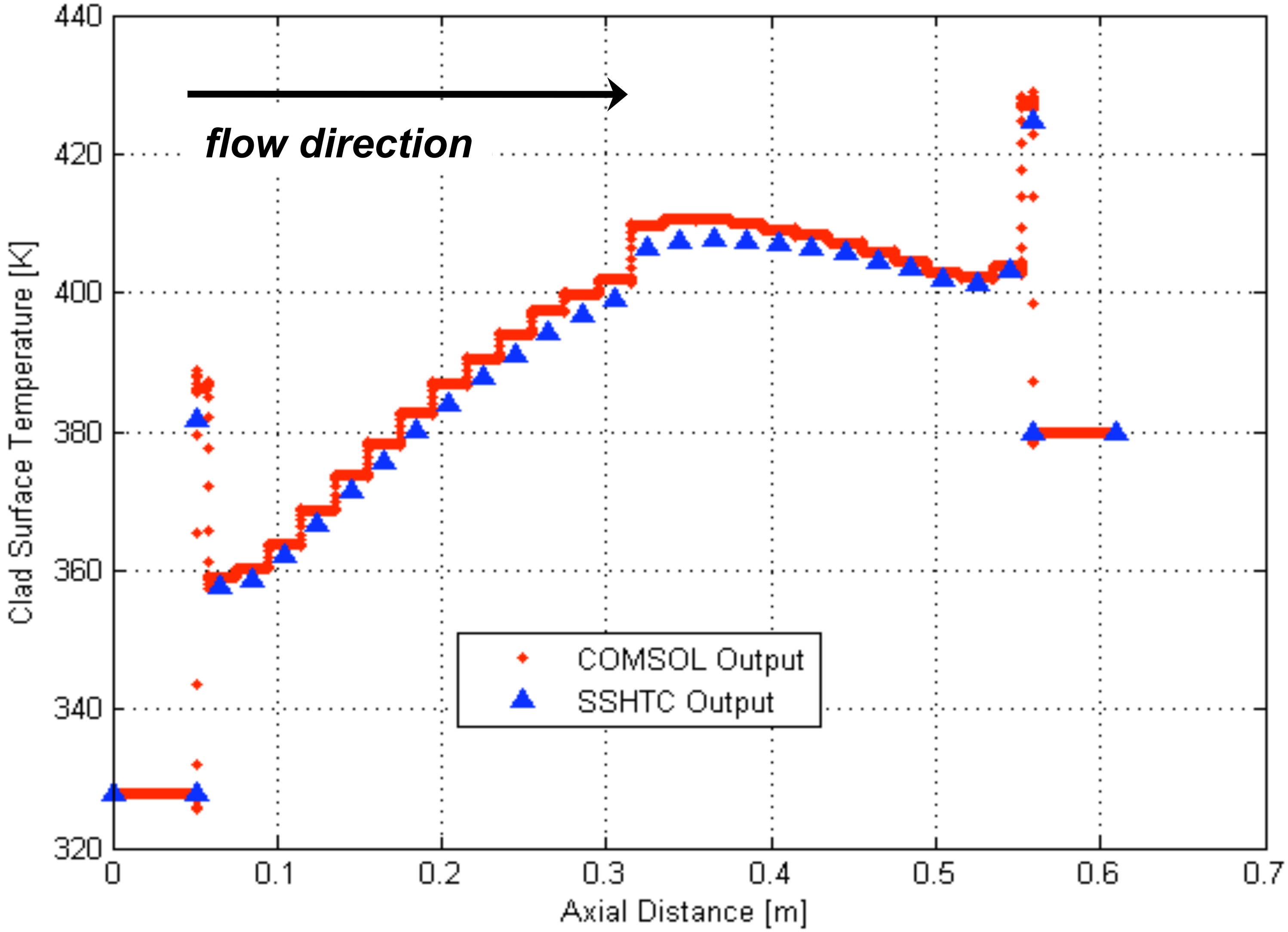
overall length is > 24” causes the mesh to be difficult to view here due to the high-aspect ratio

Use a 2D Model as a Test Bed for 3D -establish proper boundary conditions

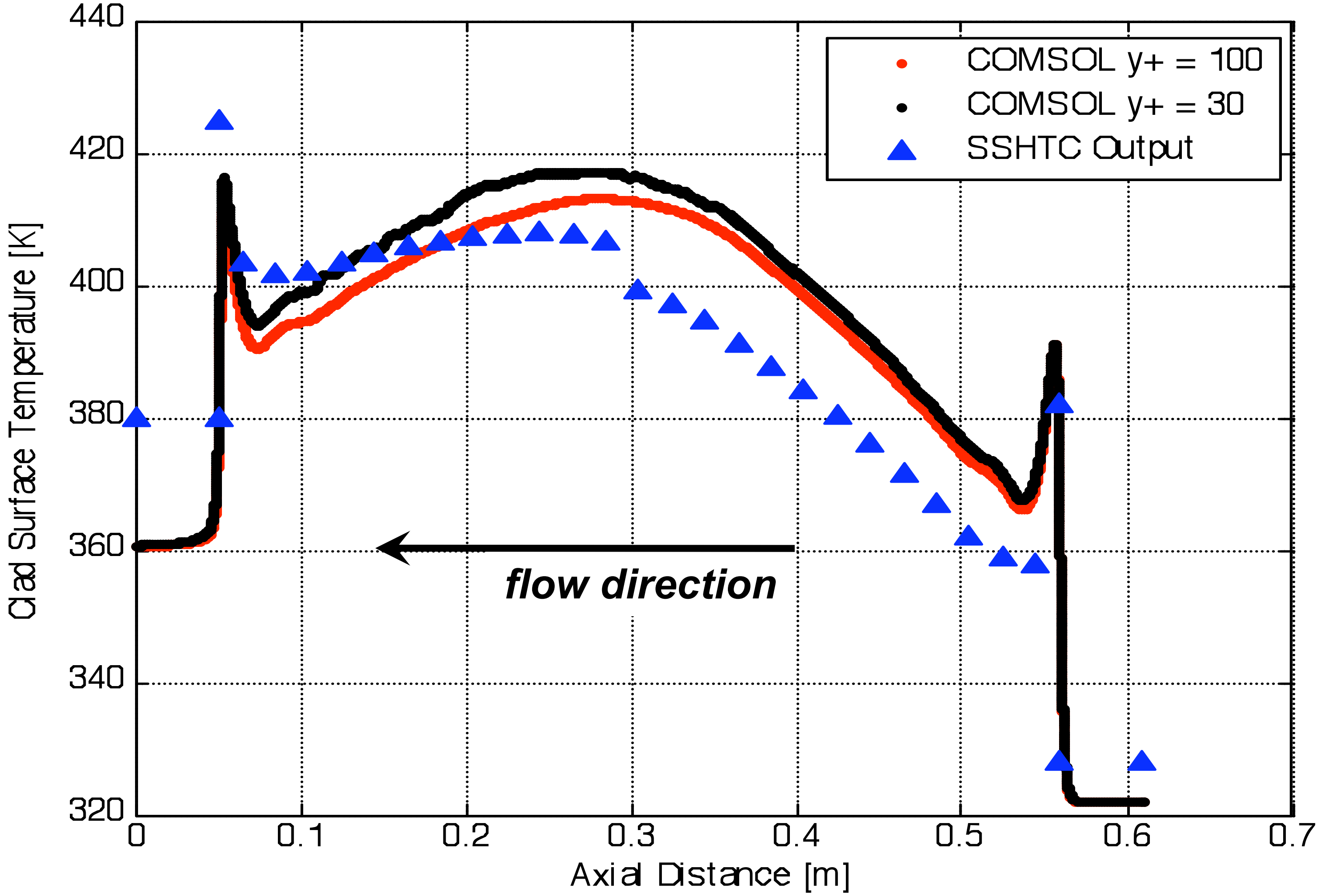


note: this is an expanded view using COMSOL post processing option “equal axis” as unchecked

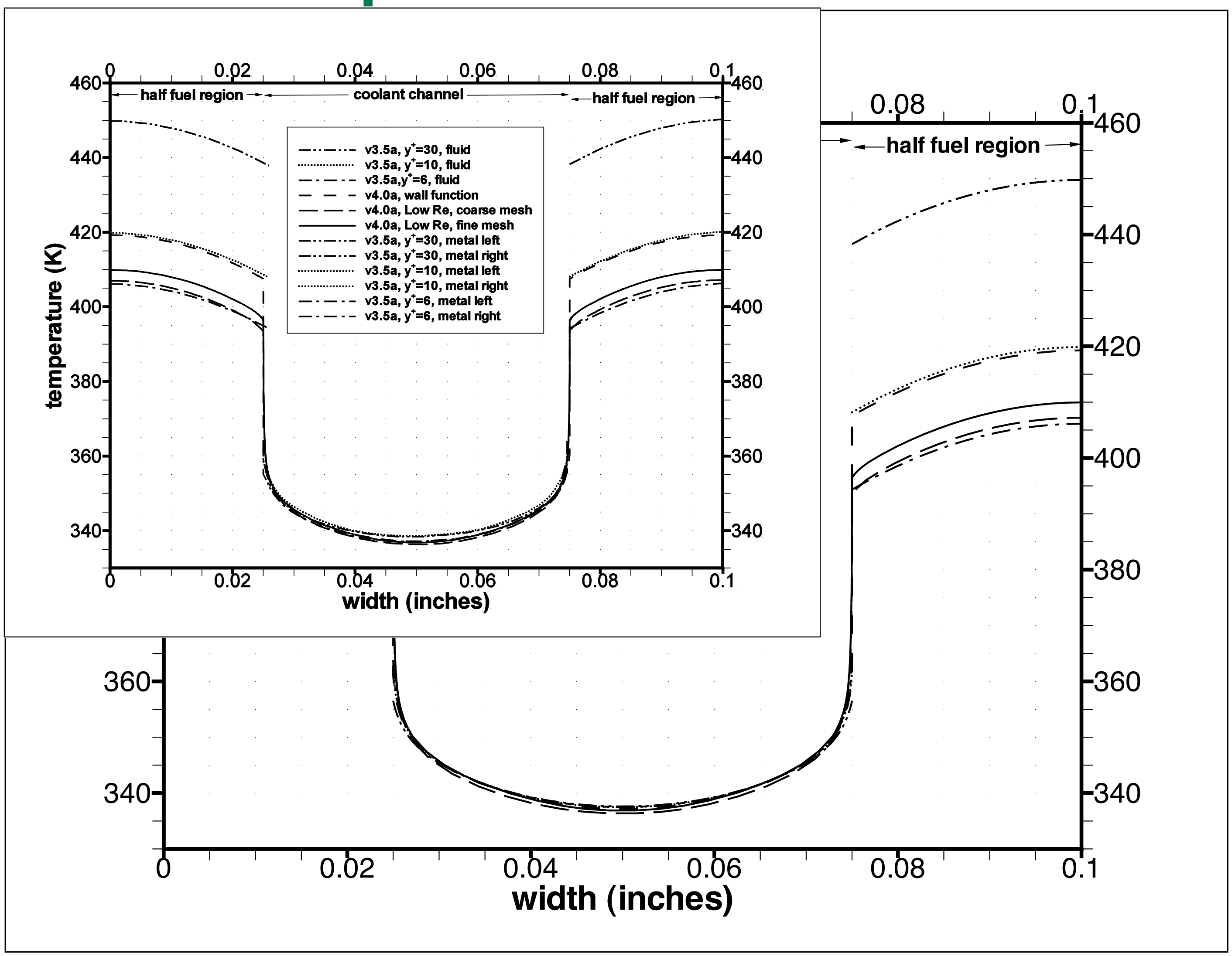
Simplifying assumptions, inputs, and boundary conditions reproduce the existing conservative safety basis (HFIR SSHTC)



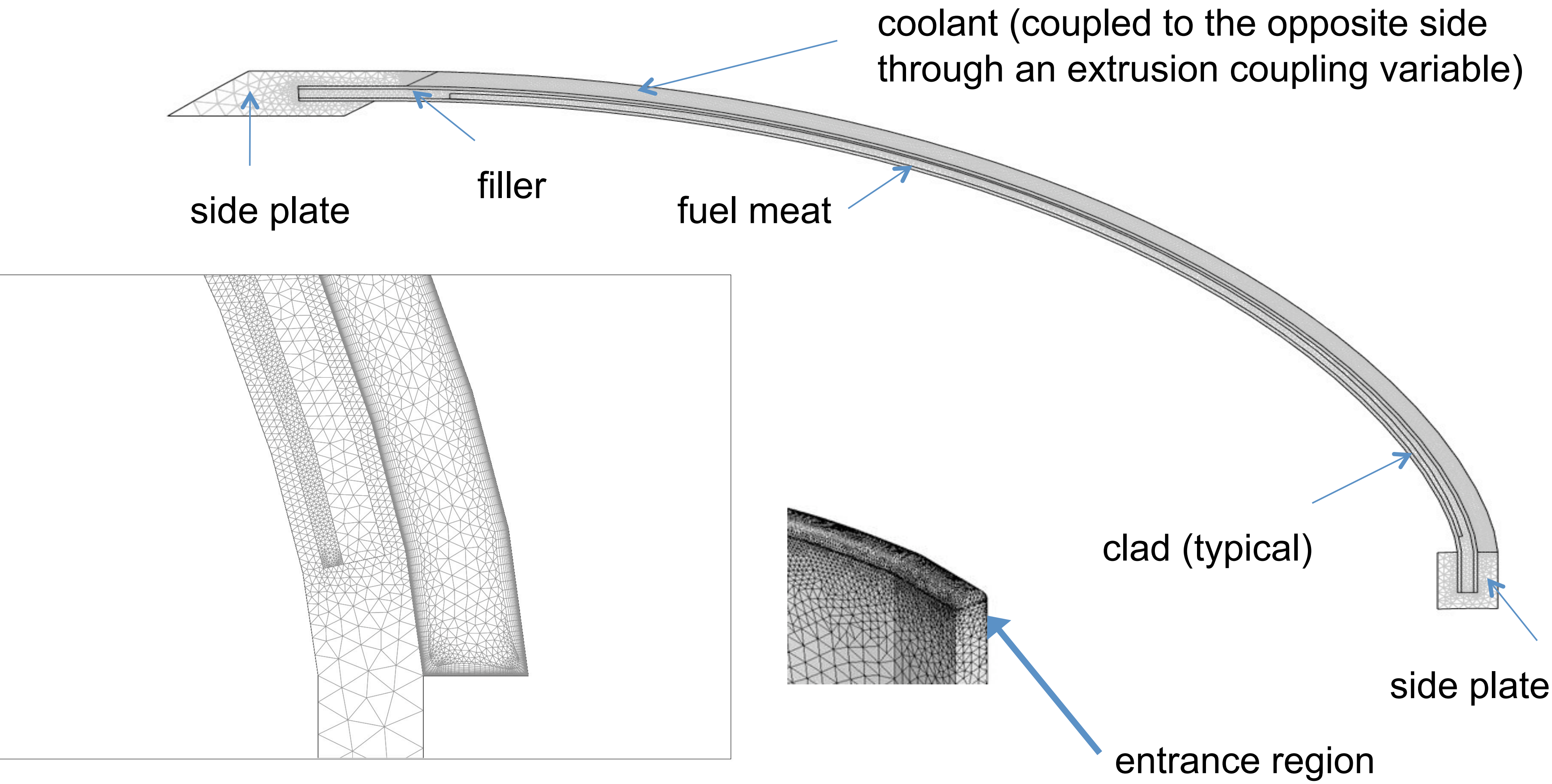
A COMSOL v3.5a best-estimate comparison with the the existing conservative safety basis provided by the HFIR SSHTC.



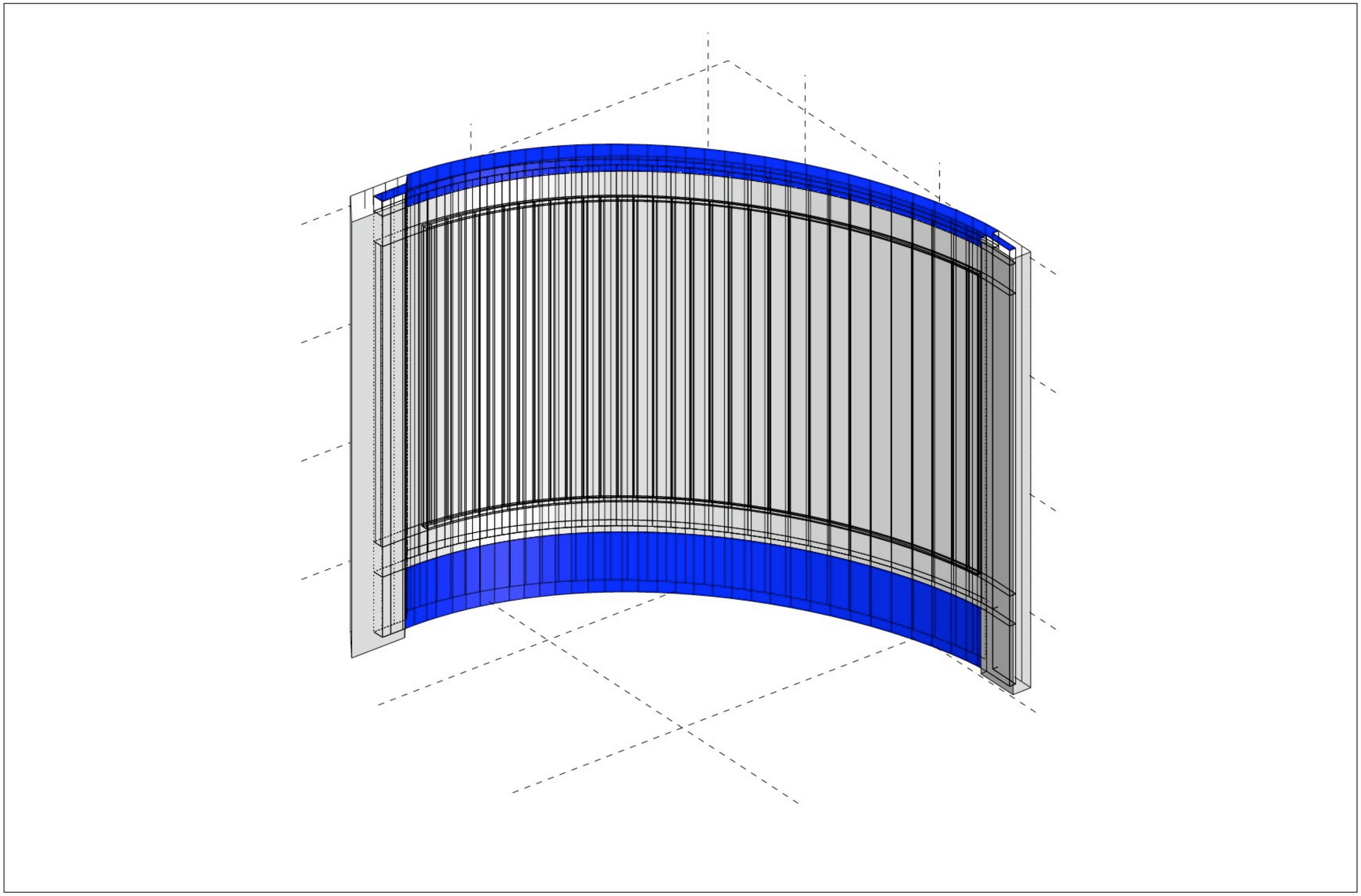
COMSOL v4.0a is providing an improved turbulence model for increased accuracy of the wall surface temperature.



3D meshing starts with a 2D working plane of the involute plate to be extruded in the axial direction

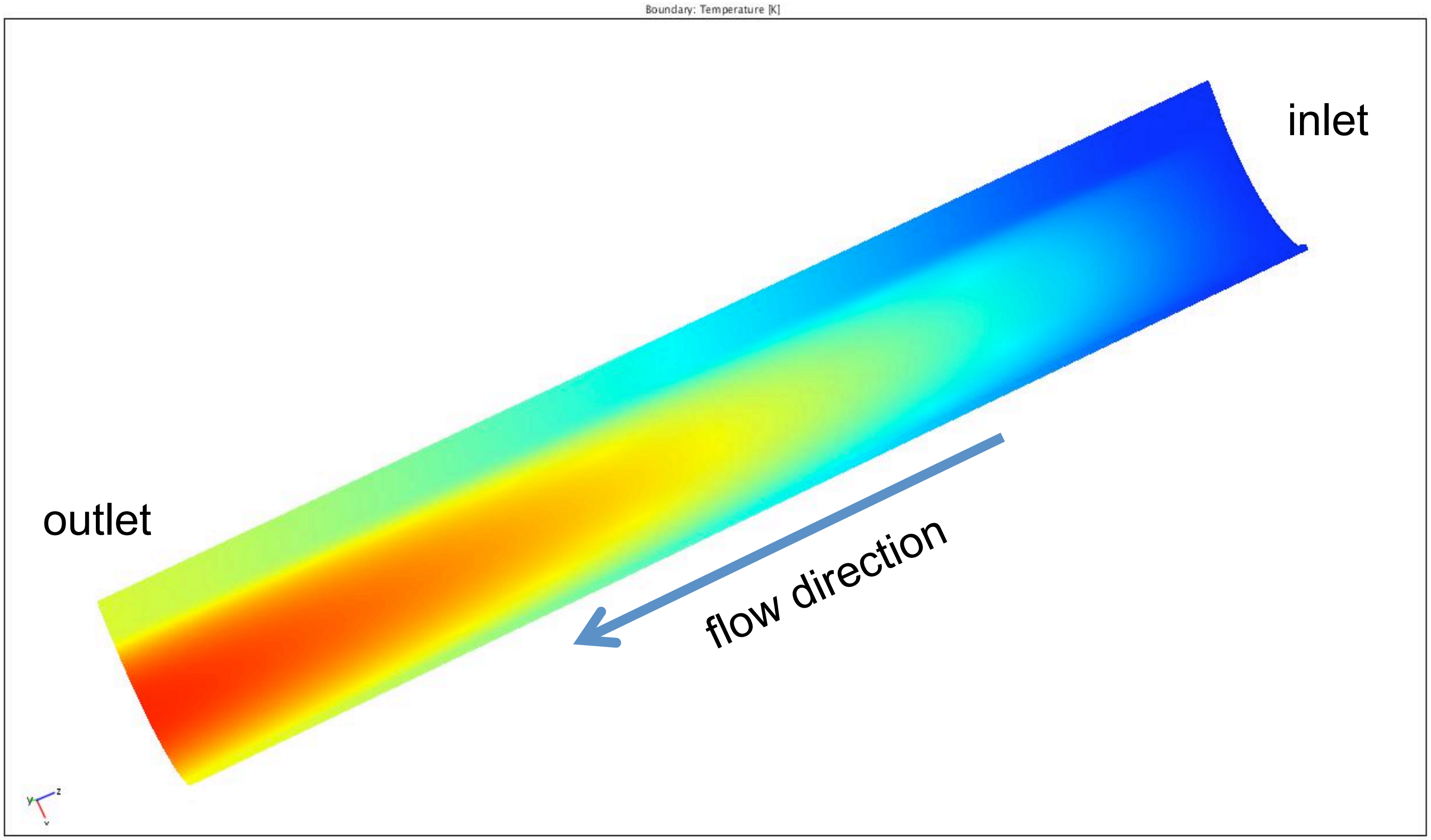


COMSOL CAD Tools Also Provide Geometry



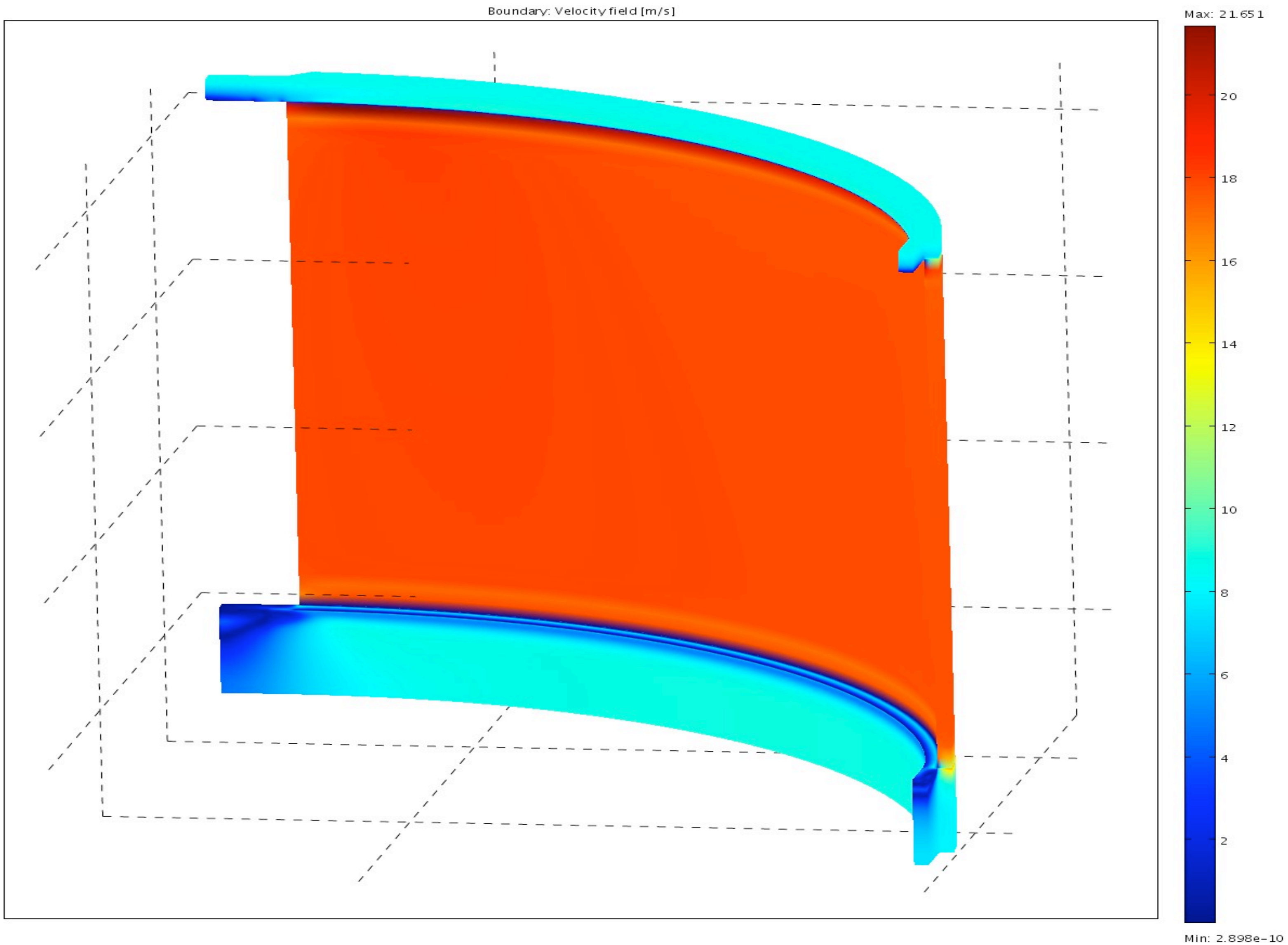
all geometry parts shown that we are modeling with COMSOL in full 3D

Typical 3D result: metal surface temperature



shown is the clad surface temperature adjacent to the turbulent-wall of the coolant

Typical 3D result: velocity distribution

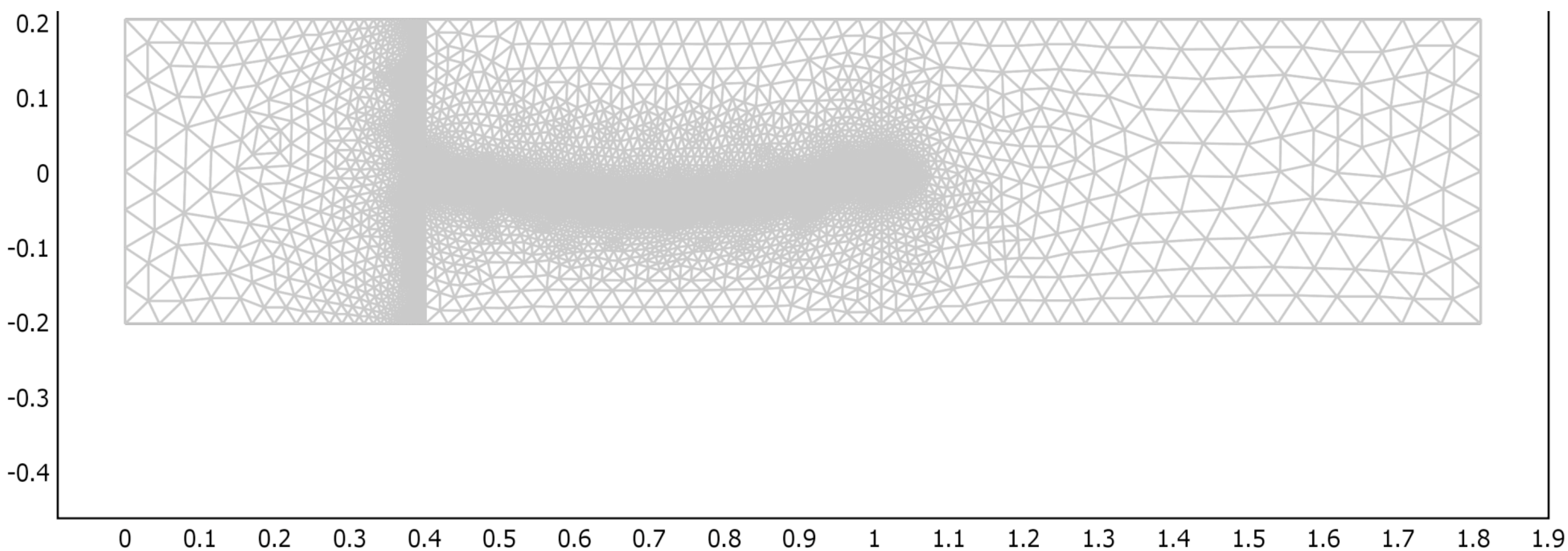


shown is the velocity distribution (not to scale, unequal axis to show details)

Structural mechanics is coupled to account for:

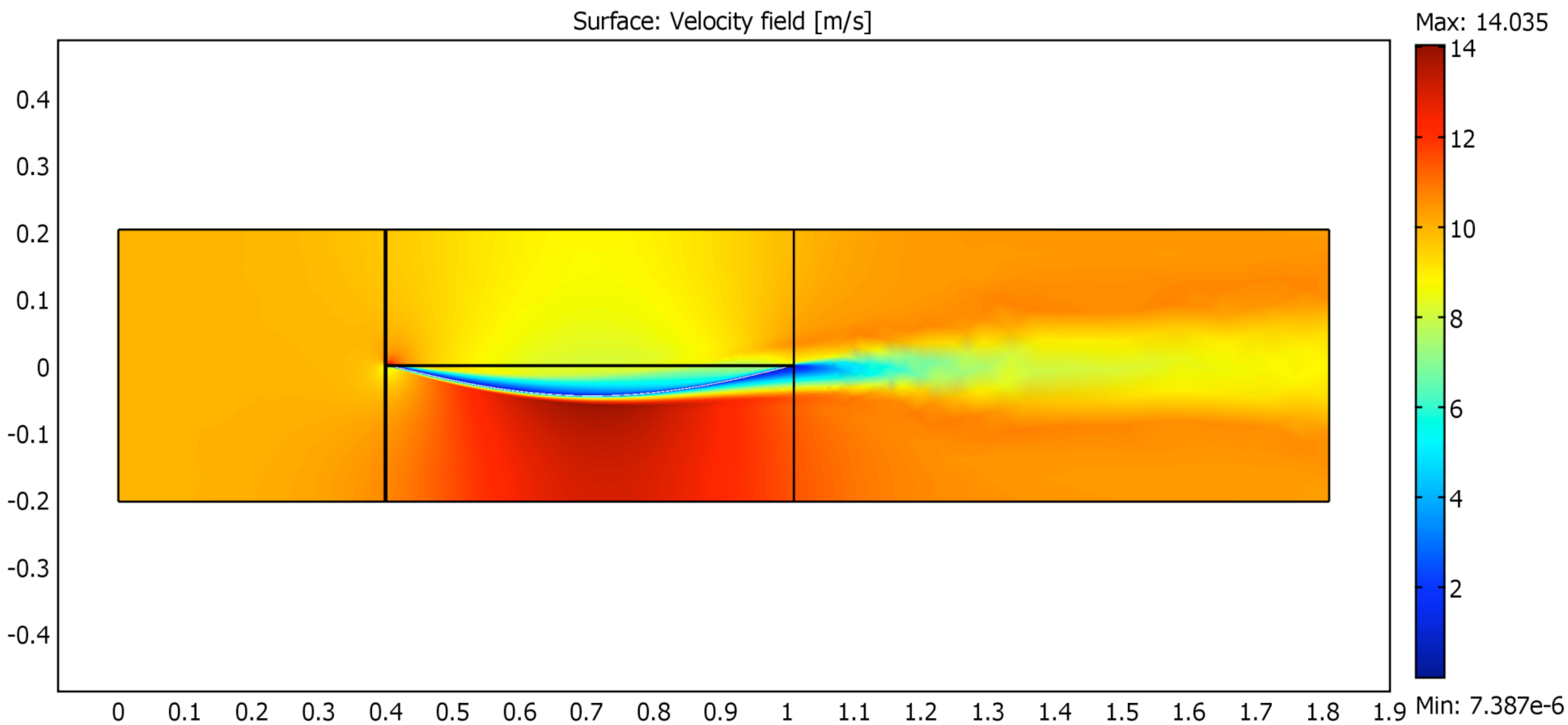
- buckling effects caused by thermal expansion

- vibrational effects caused by high-speed flow and flow changes



Structural mechanics is coupled to account for:

- buckling effects caused by thermal expansion
- vibrational effects caused by high-speed flow and flow changes



Conclusions and Current Status

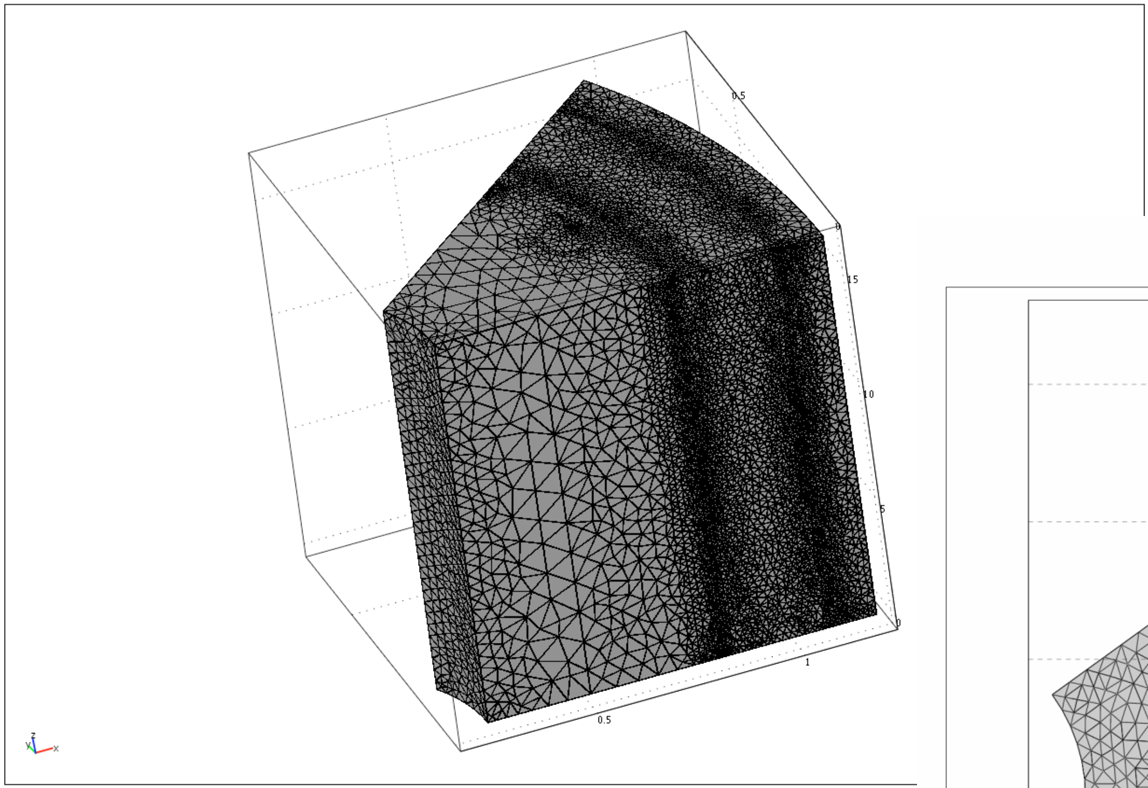
- **COMSOL is providing a modern simulation tool for the design of LEU fuel and analysis of present HEU fuel.**
- **The COMSOL analysis provides details and accuracy heretofore not possible, and allows for precise margin estimates.**
- **The 2D analysis provides a test bed for the 3D analysis to establish consistent parameter set (wall offset, mesh density, solution procedure, etc.)**
- **2D results demonstrate methods that yield accurate benchmarks with established data base.**
- **3D computational requirements have been demonstrated on a representative model.**
- **COMSOL v4.0a brings improved models and computational capability (parallel processing).**

References:

1. Freels, J. D., Bodey, I. T., Arimilli, R. V., and Lowe, K. T., “Two Dimensional Thermal Hydraulic Analysis and Benchmark in Support of HFIR LEU Conversion Using COMSOL,” Tech. Rep. ORNL/TM-2010/018, Oak Ridge National Laboratory, 2010.
2. Tschaepe, L., Ruggles, A. E., Freels, J. D., and Primm, R. T., “Evaluation of HFIR LEU Fuel Using the COMSOL Multiphysics Platform,” Tech. Rep. ORNL/TM-2008/188, Oak Ridge National Laboratory, 2008.
3. Freels, J. D., Arimilli, R. V., and Bodey, I. T., “Exploiting New Features of COMSOL Version 4 on Conjugate Heat Transfer Problems,” *COMSOL Conference Boston 2010 (available on CD).*
4. Bodey, I. T., Arimilli, R. V., and Freels, J. D., “Upgrading the HFIR Thermal-Hydraulic Legacy Code Using COMSOL, ” *COMSOL Conference Boston 2010, (available on CD).*
5. Freels, J. D., Arimilli, R. V., Lowe, K. T., and Bodey, I. T., “The Role of COMSOL Toward a Low-Enriched Uranium Fuel Design for the High Flux Isotope Reactor,” C*OMSOL Conference Boston 2009 (available on CD).*
6. Freels, J. D., “Implementation of Oak Ridge National Laboratory Software Quality Assurance Requirements for COMSOL 3.4,” *COMSOL Conference Boston 2008 (available on CD)*.
7. Freels, J. D., “COMSOL Validation Progress on Supercritical Hydrogen Heat Transfer,” *COMSOL* Conference Boston 2007 (available on CD).
8. Freels, J. D., “Influence of COMSOL on the Design and Testing of the High Flux Isotope Reactor HB-4 Cold Source: Validation of the Simulation,” *COMSOL Conference* Boston 2006, October 2006, invited keynote talk (presentation only, available on CD).
9. Freels, J. D., “Application of FEMLAB on Supercritical Hydrogen Components of the High Flux Isotope Reactor Cold Source,” *COMSOL Conference Boston 2005 (available on CD)*.

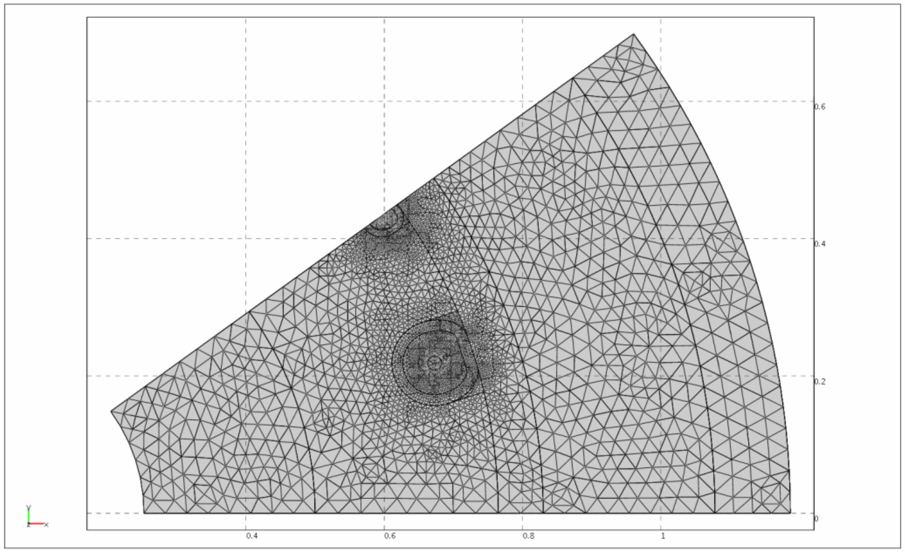
Extra Slides

Success Story: COMSOL simulations of the cold source hydrogen test heater (part 1 of 2)

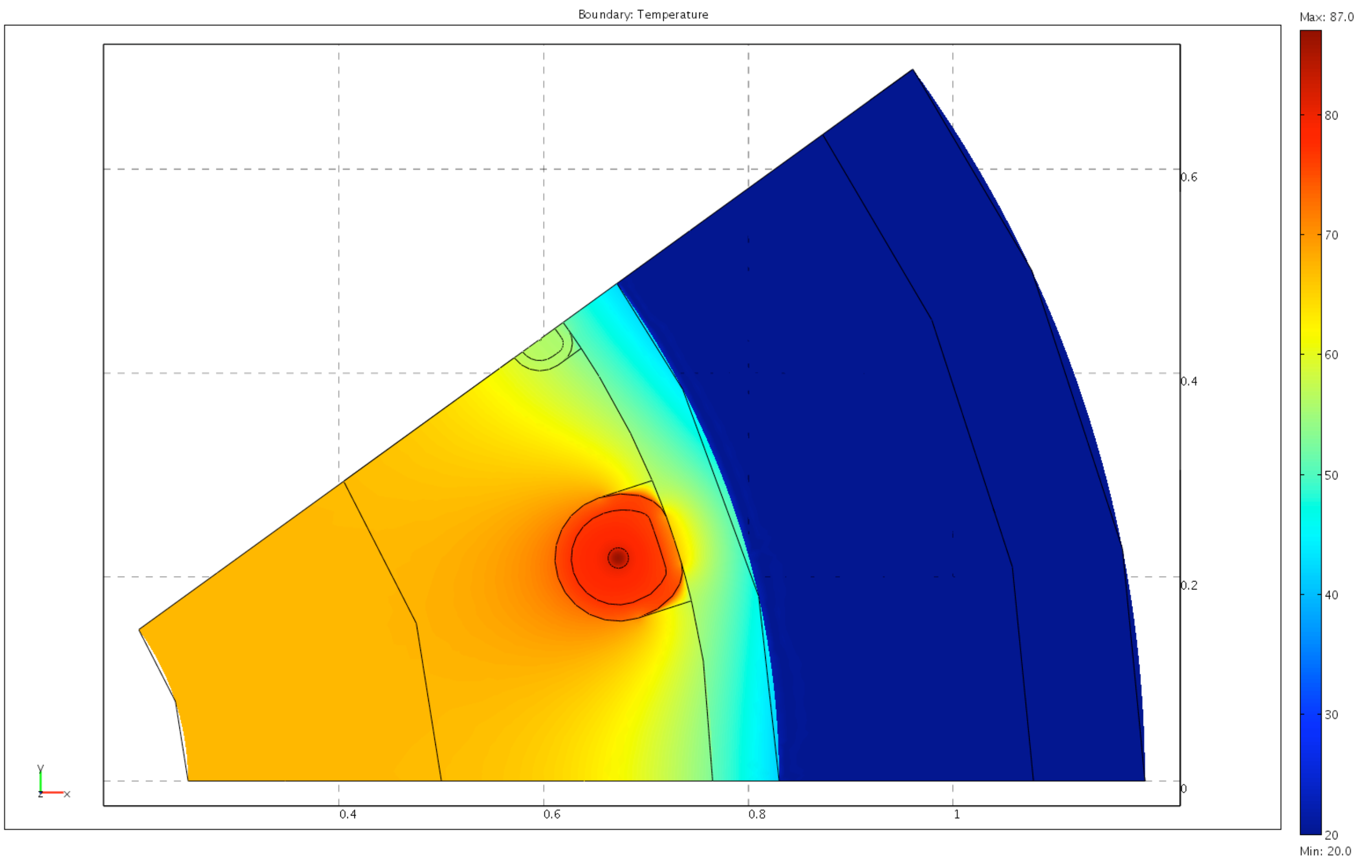


Perspective view

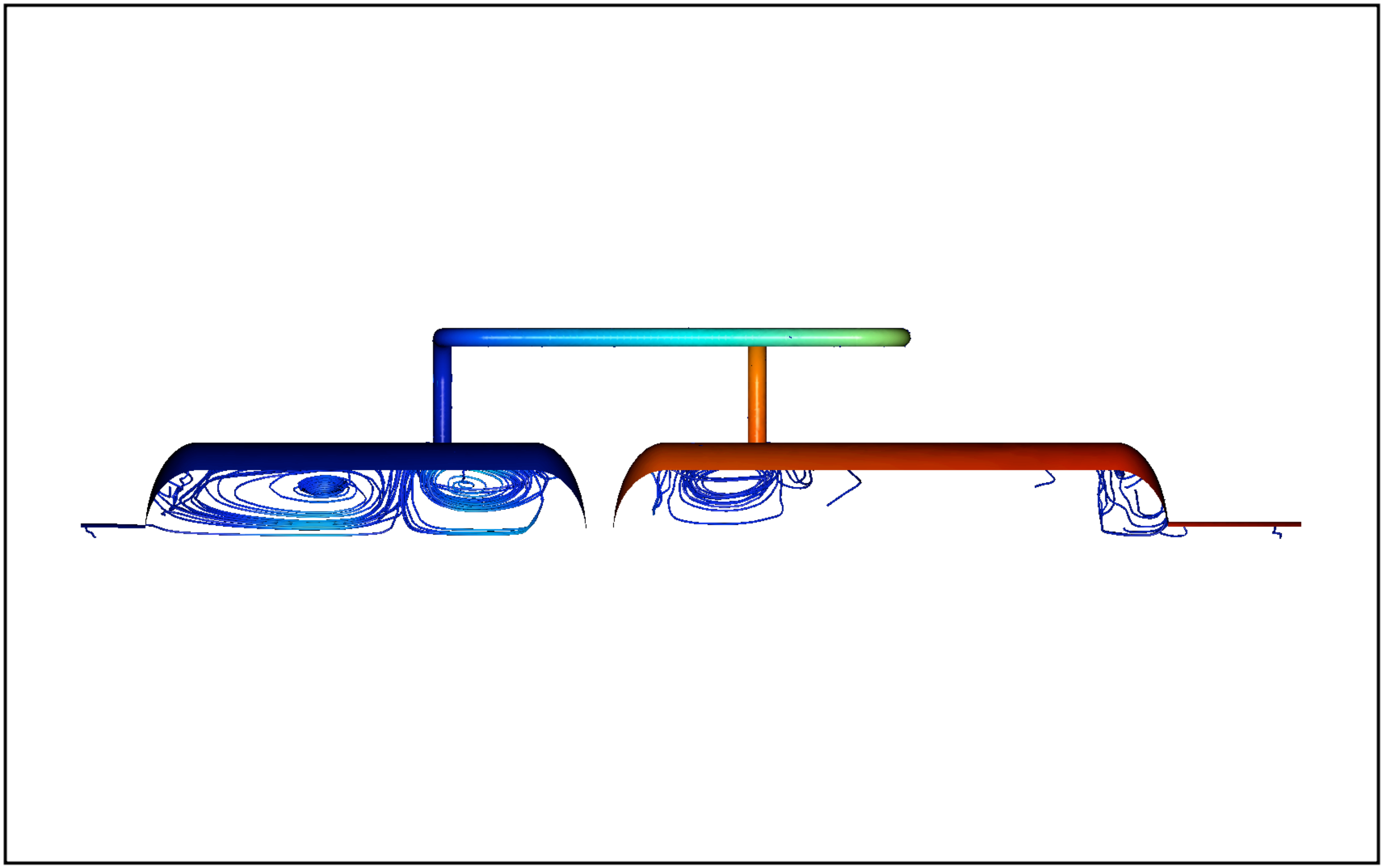
Top view



Success Story: COMSOL simulations of the cold source hydrogen test heater (part 2 of 2)



Success Story: COMSOL multiphysics simulation of the HFIR cold source pressurizer Part 2 of 3



Input physics specifications:

- full power and flow

- Re ~ $4x10^4$ based on channel width of 0.050”

- Kays-Crawford expression for Pr_T used throughout

- V_in=7.945 m/s (26.07 ft/s), T_in=321.9 K (120 F), q’’’=$5.316x10^9$ W/m^3

- core exit pressure set to ~ 373 psia for water property evaluation

- solid properties are based on RELAP5 HFIR model and vary with temperature

-water properties are based on NIST/ASME Database 10 v2.21 and allowed to vary with temperature and pressure

- investigated y^+=30, 100, and 10 (and as related to boundary-layer mesh spacing at the wall) ***: graphics shown here for y^+=10***.

- investigated linear and quadratic finite-element basis

- investigated triangles and quadrilateral finite-elements ***: graphics shown here for triangles***.

2D solutions: COMSOL options

- k-ε turbulent, non-isothermal (weakly compressible) model

- GLS dissipation on Navier-Stokes, turbulence, and coolant temperature

- cross-flow dissipation on Navier-Stokes, and coolant temperature

- NO additional isotropic dissipation used !

- realizability constraint on turbulence model is used

- corner smoothing *not* used in 2D solutions, may be needed in 3D

- included optional surface heating from pressure and viscosity terms

- use manual scaling in advanced solution options

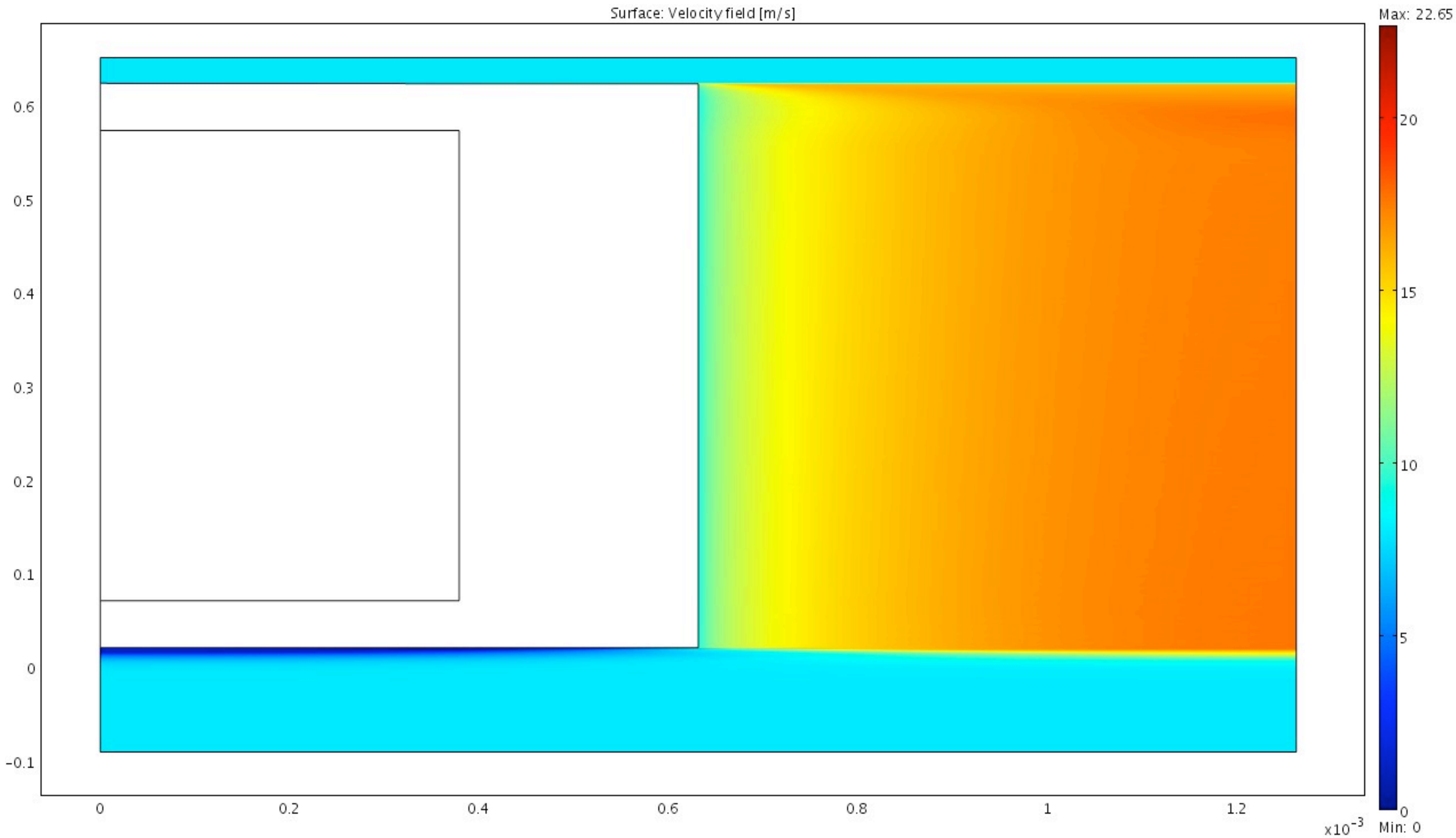
- use parametric solver on power level to get to full power

- used segregated solver to get to final solution

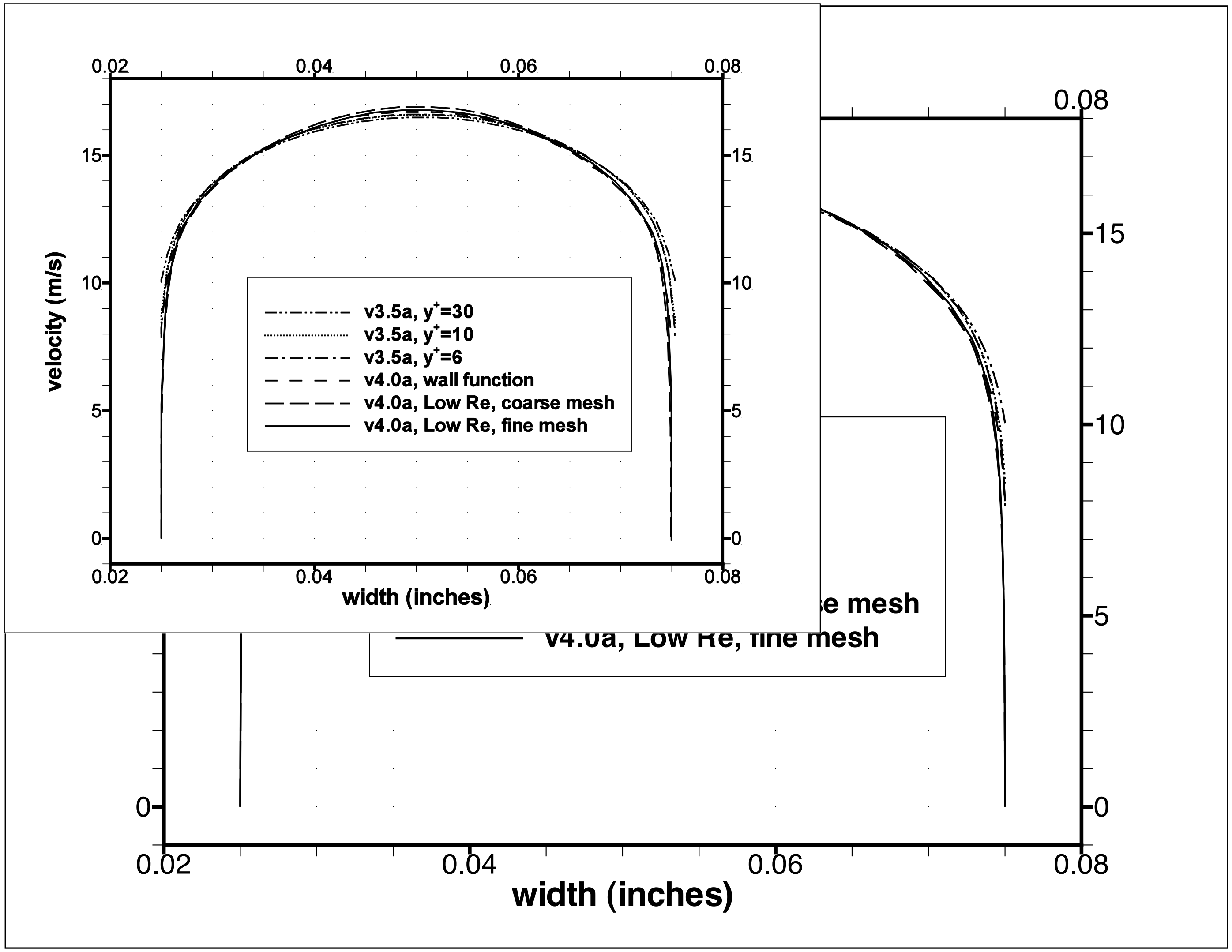
- final 2D solution is fully-coupled direct solver

- iterative solver in 3D (see full paper for discussion)

2D solutions: U_chns, quad elements, 4671216 dof



COMSOL v4.0a is providing an improved turbulence model for increased accuracy of the wall surface temperature (part 1 of 2).



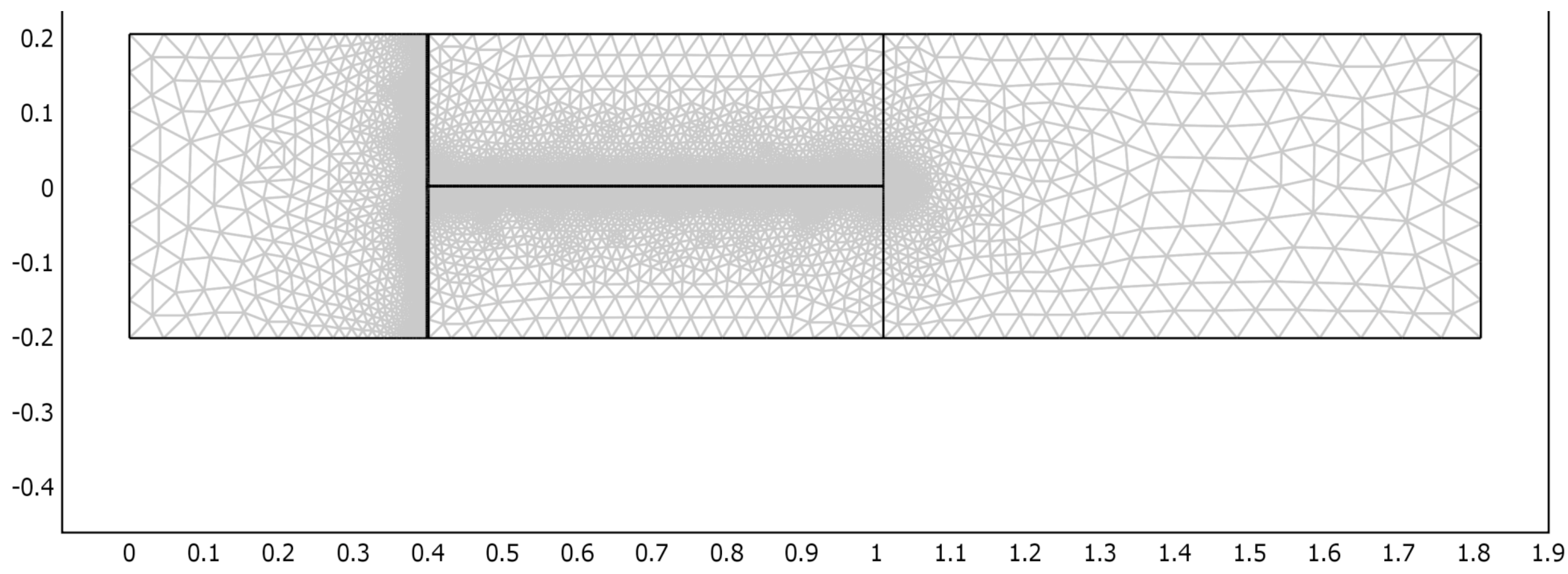
Involute plate geometry yields an asymmetry of the coolant channel at the element sidewalls

- **The 3D model was expanded to include a full single flow channel and a full fuel plate on one side**
- **A COMSOL extrusion coupling variable allows for this configuration to be used**
- **This option was investigated along with several additional options**
- **Most important is the asymmetry of the fuel plate itself due to the non-symmetric fuel loading**
- **Also investigating Solidworks interface for more consistent user interface to the boundary conditions**
- **A final configuration has been established for detailed, best-estimate single plate model of the HFIR**

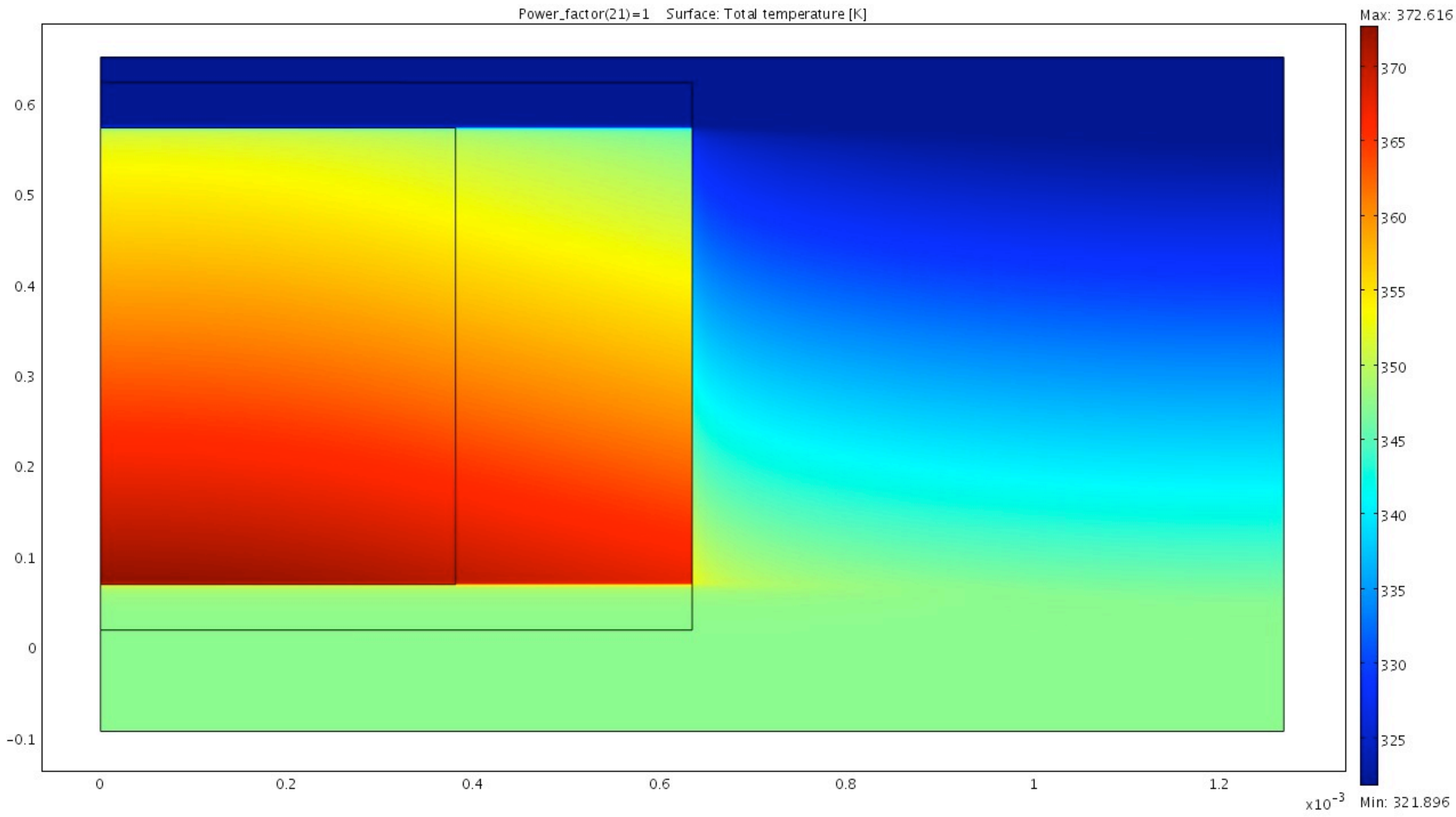
Structural mechanics is coupled to account for:

- buckling effects caused by thermal expansion

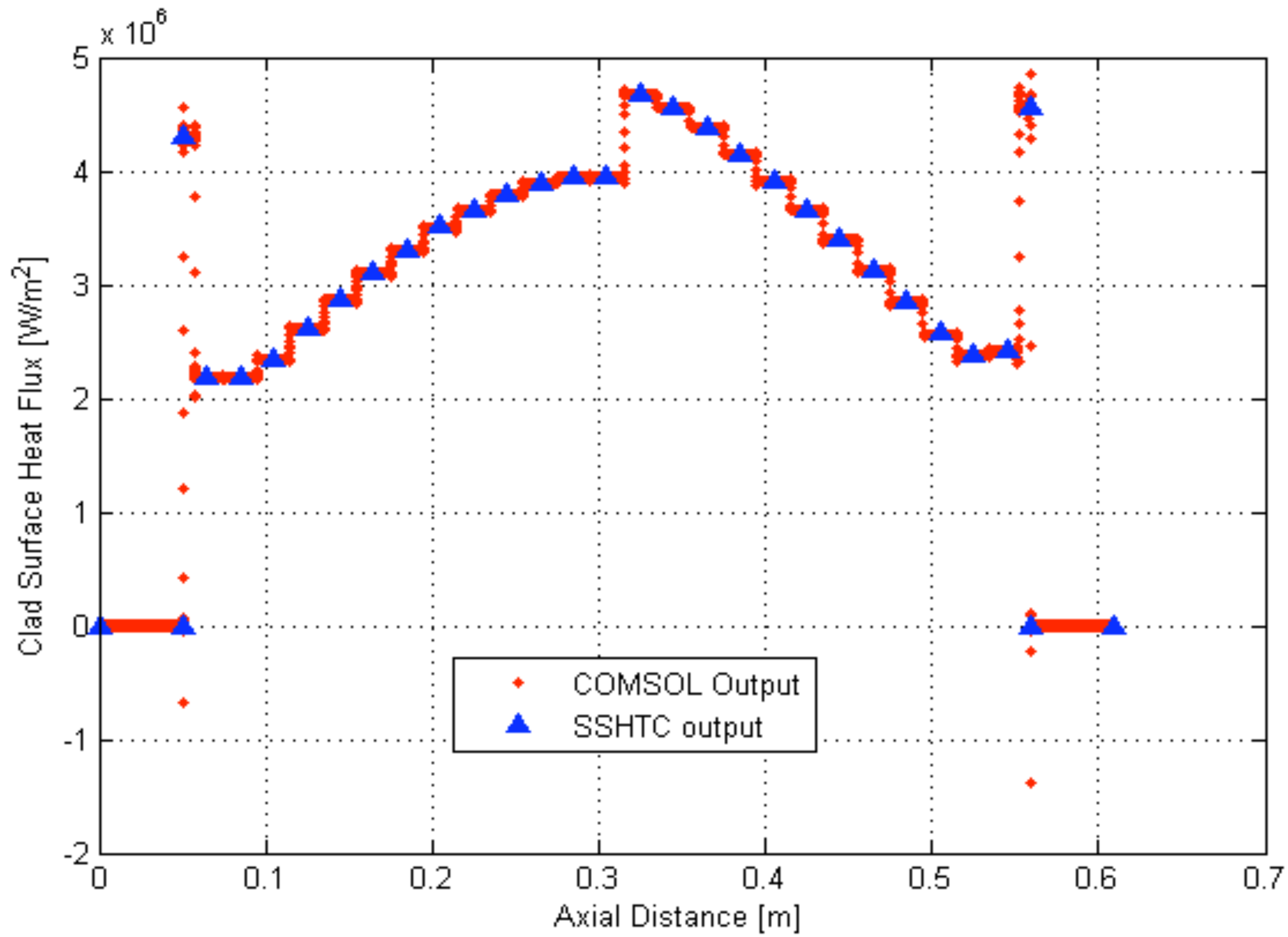
- vibrational effects caused by high-speed flow and flow changes



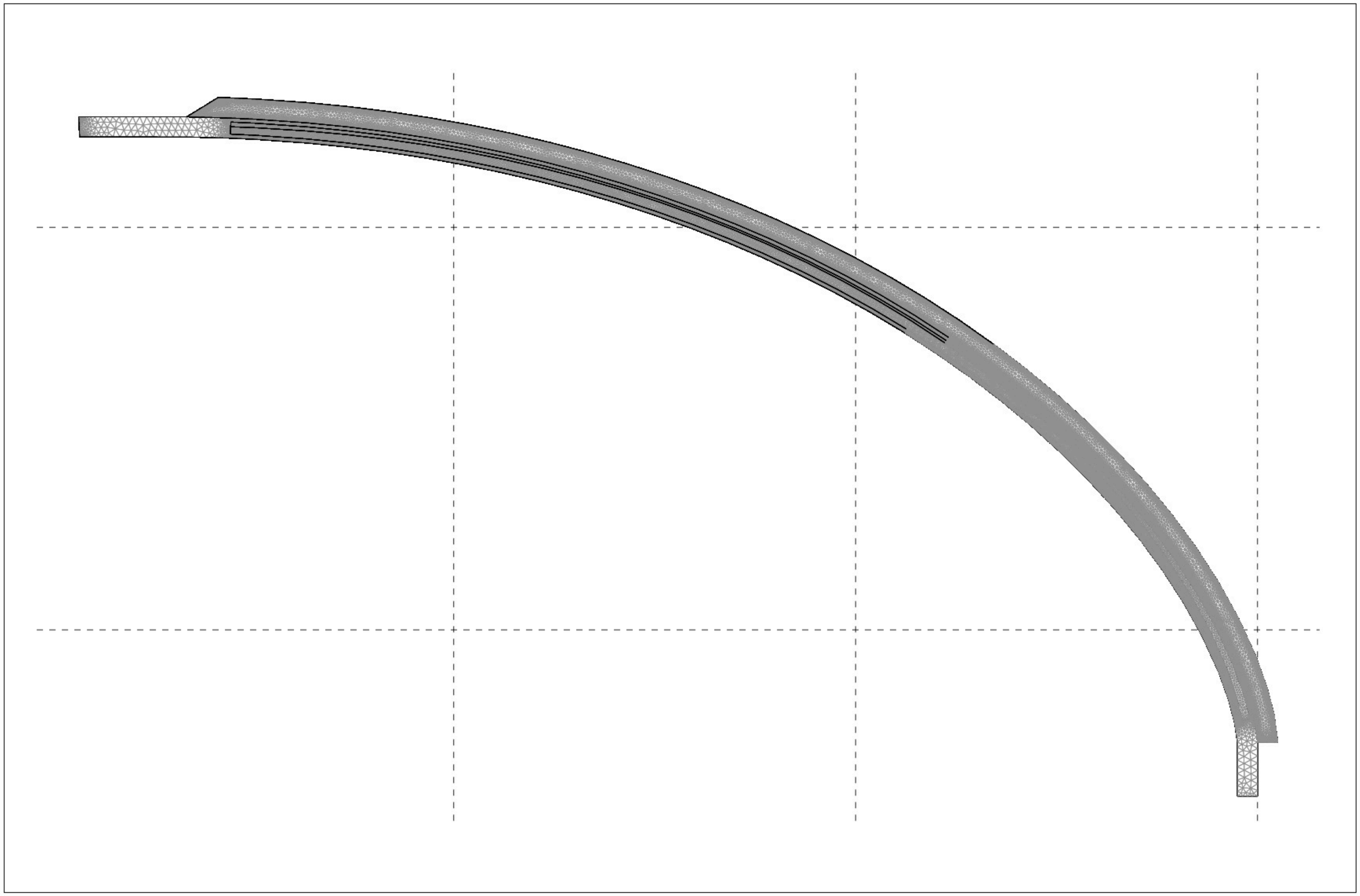
2D solutions: Ttot, linear elements, 1346998 dof



Simplifying Assumptions, Inputs, and Boundary Conditions can Reproduce the Existing Conservative Safety Basis (HFIR SSHTC, part 1 of 2)

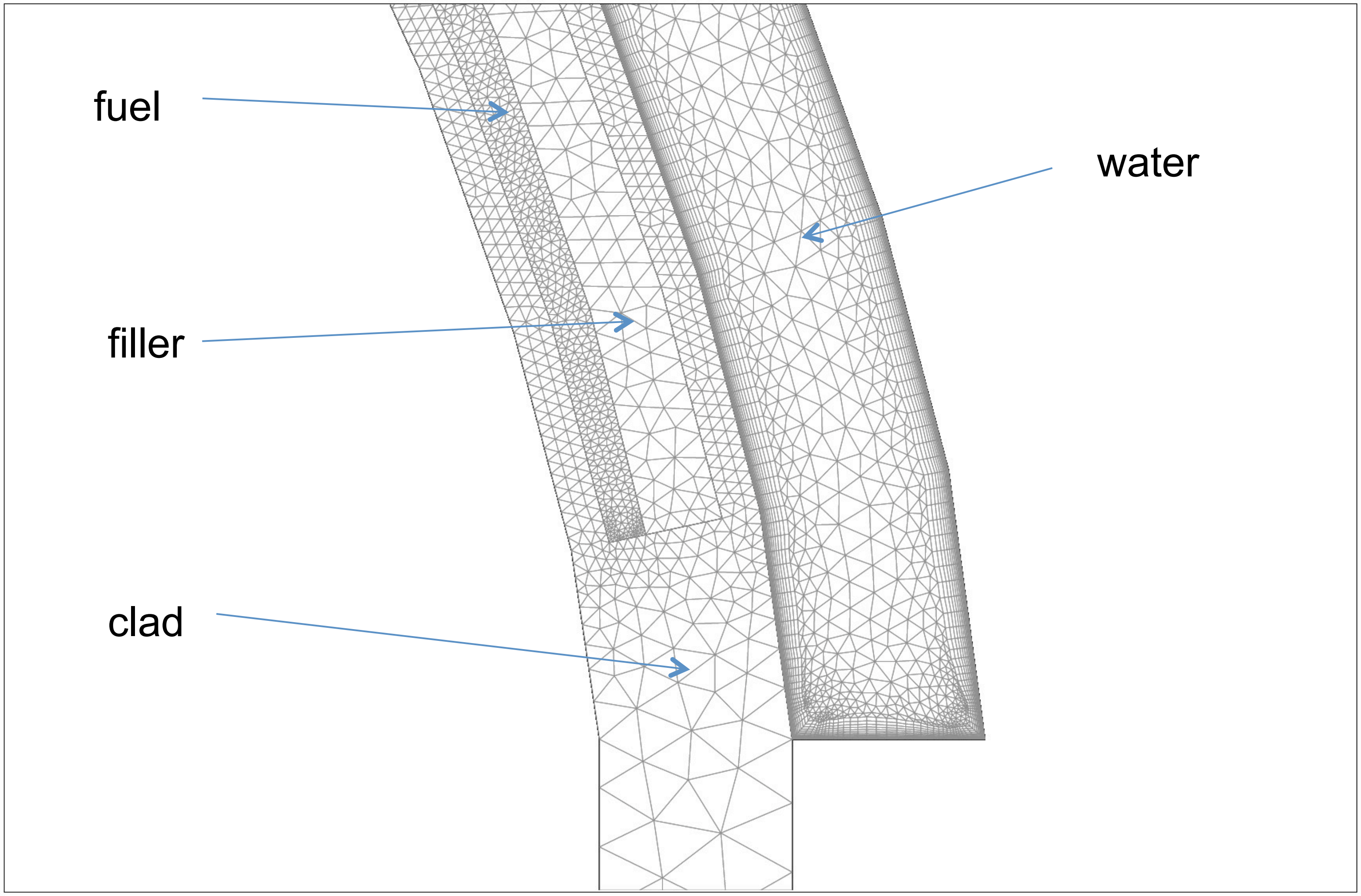


COMSOL Meshing Tools Yield High Quality



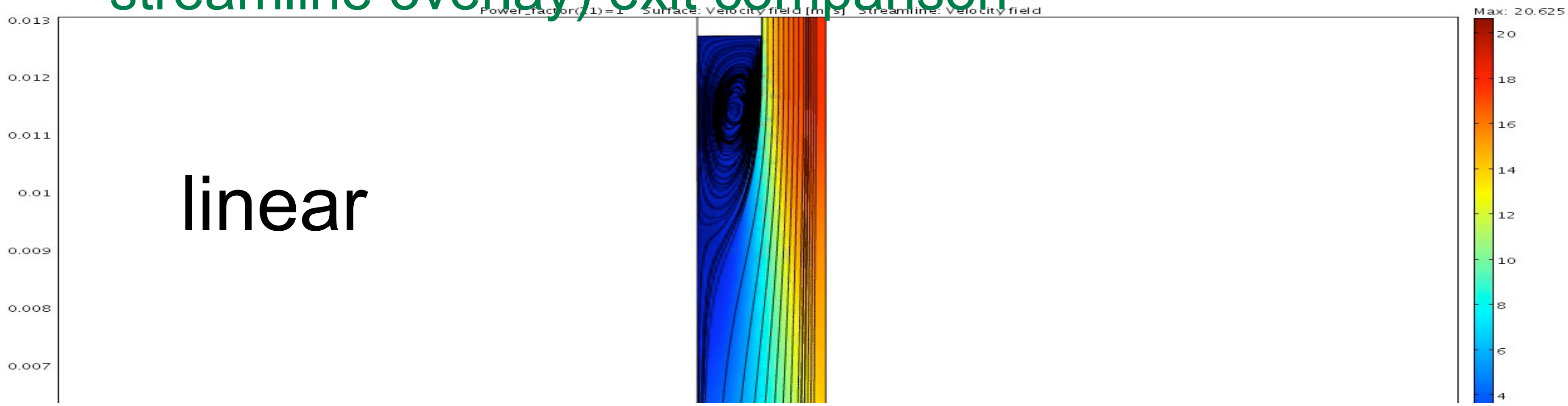
coolant and fuel region meshing top view, excellent mesh !

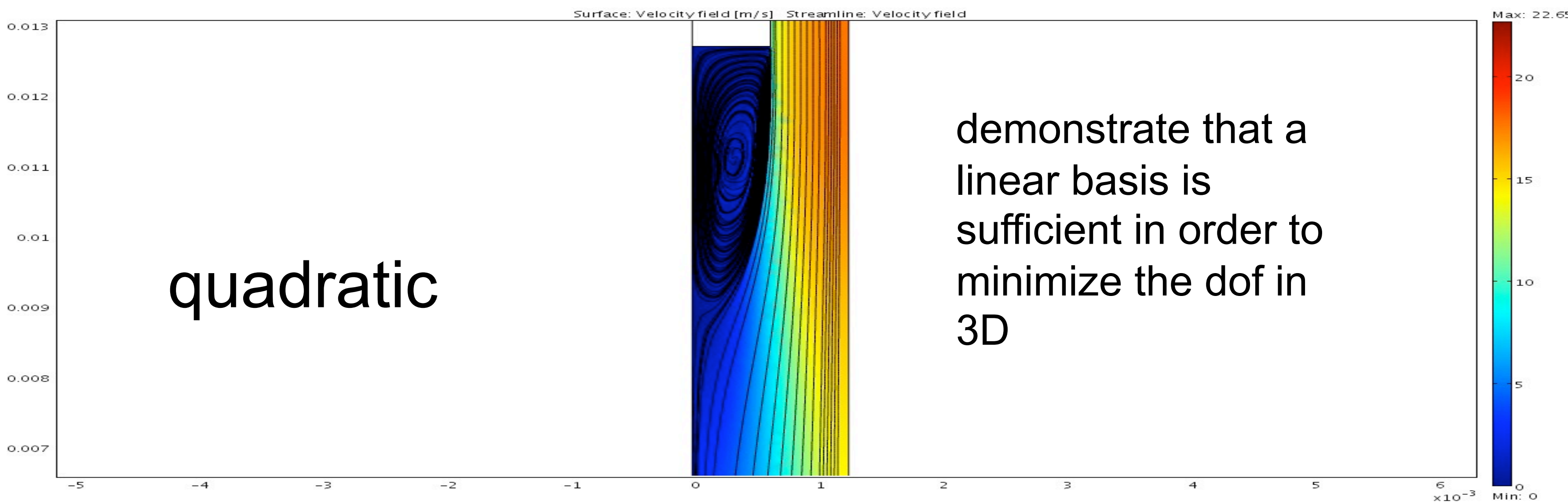
COMSOL Meshing Tools Yield High Quality



coolant and fuel region meshing top view magnified, note parts meshed

2D solutions: U_chns (velocity magnitude with streamline overlay) exit comparison





demonstrate that a linear basis is sufficient in order to minimize the dof in 3D

Typical 3D issue: creating accurate geometry

This model of a solid aluminum plate was created using Solidworks CAD software. Note: rounded top is difficult to achieve using CAD tools provided with COMSOL. However Solidworks has difficulty providing an accurate representation of the plate internals (fuel and filler) that is mesh-able with COMSOL.