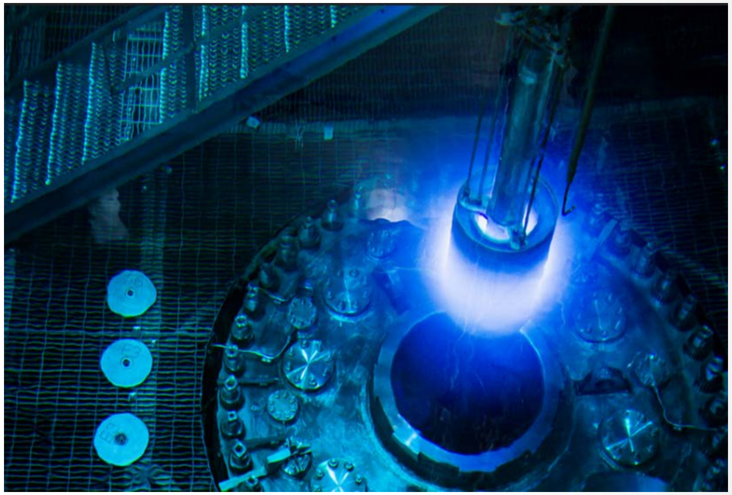
Multiphysics Simulation using high resolution FSI Modeling to Support Safety and Reliability of New HFIR Fuel at ORNL

Dr. Amir I. Elzawawy
Vaughn College of Aeronautics and Technology
Dr. James D. Freels
Oak Ridge National Laboratory
Franklin Curtis
Oak Ridge National Laboratory



Presented by:
Amir Elzawawy
Vaughn College of Aeronautics & Technology

Motivations

- Global Threat Reduction Initiative (GTRI) to use Low Enriched Uranium (LEU) instead of High Enriched Uranium (HEU) at research reactors.
- Create a full physical model to simulate HFIR physics including Thermal hydraulics, nuclear physics

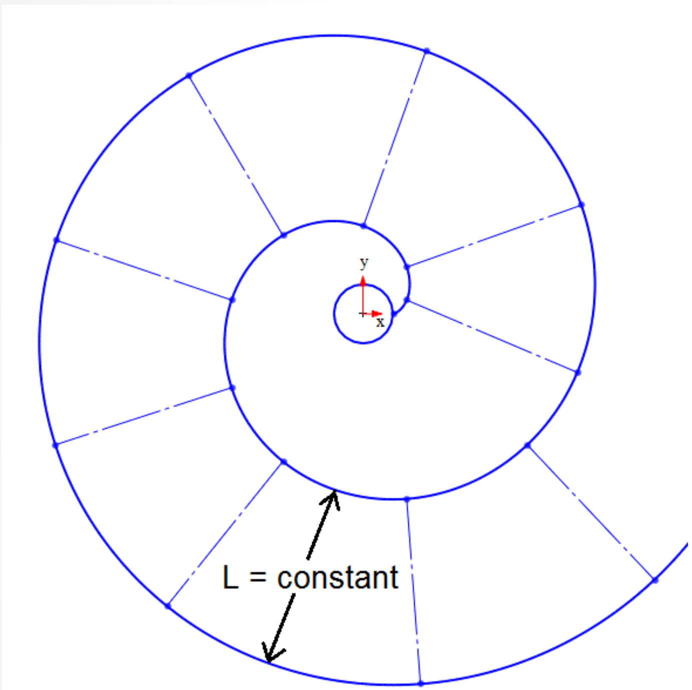
Objective

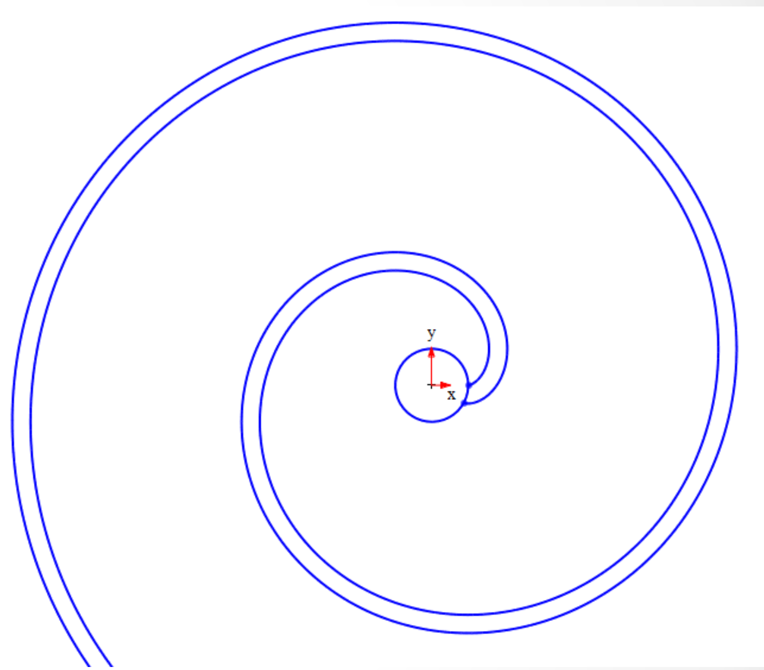
- High resolution FSI (Fluid-Stricture Interaction) modeling for the cooling process of the fuel plates.

Fuel Plates

- 171 Inner Fuel Elements and 369 Outer Fuel Elements
- Nominal Fuel thickness = 0.0050 inch
- Gaps between each two elements = 0.0050 inch

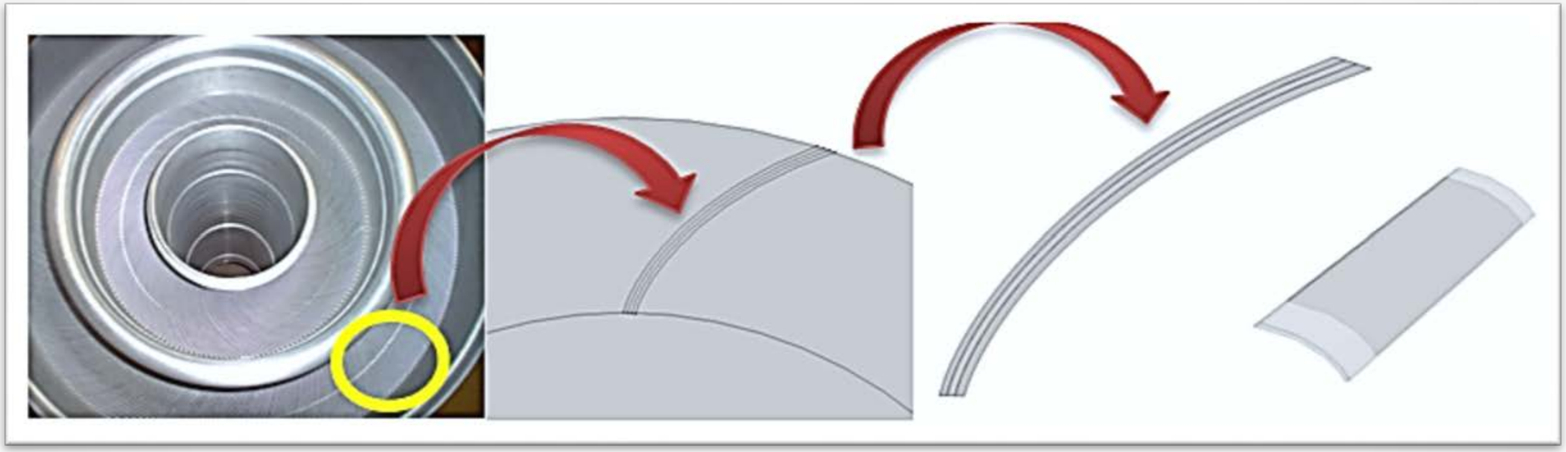
Involute-Shaped Fuel Elements





- Channel Size maintained constant and independent of the radial direction.
- Allows water to flow between the fuel plates for cooling while reducing the three-dimensional flow effect.

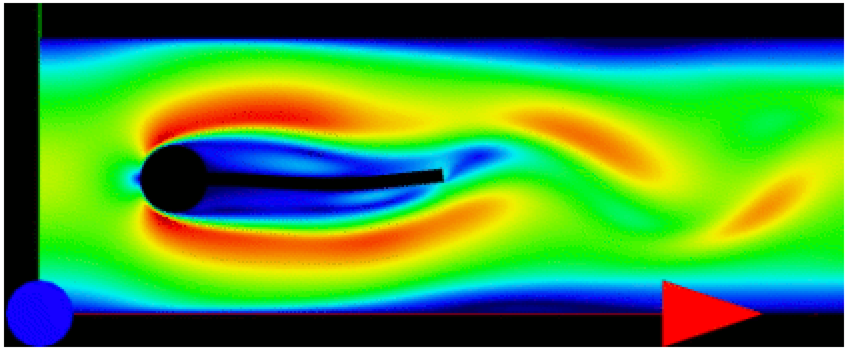
Modeling of HFIR Fuel Plate with cooling water



- Due to the azimuthal symmetry of the HFIR fuel elements, a single fuel plate and the two adjacent coolant channels of the outer HFIR fuel element is modeled.

Fluid-Structure Interaction (FSI)

- The model is intended to capture simultaneous interaction between solid displacement due to flow loadings and the change of the flow characteristics due to displacement/deformation in the solid structure.



Source: http://www.cfdanalysisservices.com/fluid-structure-interaction.php

- Physics Solved: 1- Structure Mechanics, 2- Fluid Dynamics, 3- Moving Mesh

Physics Solved:

1- Structure Mechanics

$$-\nabla \cdot \sigma = F_V$$

2- Fluid Dynamics

Turbulent Flow: Reynold's Average Navier-Stokes (RANS) K-ε Model

$$\rho(\mathbf{u}_{\text{fluid}} \cdot \nabla)\mathbf{u}_{\text{fluid}} = \nabla \cdot \left[-p\mathbf{I} + \mu\left(\nabla \mathbf{u}_{\text{fluid}} + (\nabla \mathbf{u}_{\text{fluid}})^T\right) - \frac{2}{3}\mu(\nabla \cdot \mathbf{u}_{\text{fluid}})\mathbf{I}\right] + \mathbf{F}$$

$$\nabla \cdot (\rho \mathbf{u}_{\text{fluid}}) = 0$$

3- Moving Mesh

Updates the mesh following the displacement in the fluid and the structure domains.

Segregated Solvers

- Applicable to weakly coupled problems.
- Strongly non-linear problems will suffer from slow or no convergence.
- Use to solve iteratively between different solution variables.
- Saves memory.
- Can be very efficient.

Fully Coupled Solver

- Applicable to all coupled nonlinear problems.
- Must be used with strongly coupled physics.
- Convergence more likely to be reached.
- Very expensive
- Example:

-Flexible material under heavy fluid loading

STEP-1: FSI COMSOL Solutions- Comparison between Segregated and Fully Coupled Solutions

Objective of STEP-1: is investigate the outcome of the solution based on both models.

- **Segregated Model ➡ separately solves each physics in a loop ➡ less computer memory (RAM)**
- **Fully Coupled Model ➡ solves all the physics at the same time ➡ More memory is required**

STEP-1: FSI COMSOL Solutions- Comparison between Segregated and Fully Coupled Solutions

One-Way Coupling (Fluid → Solid)

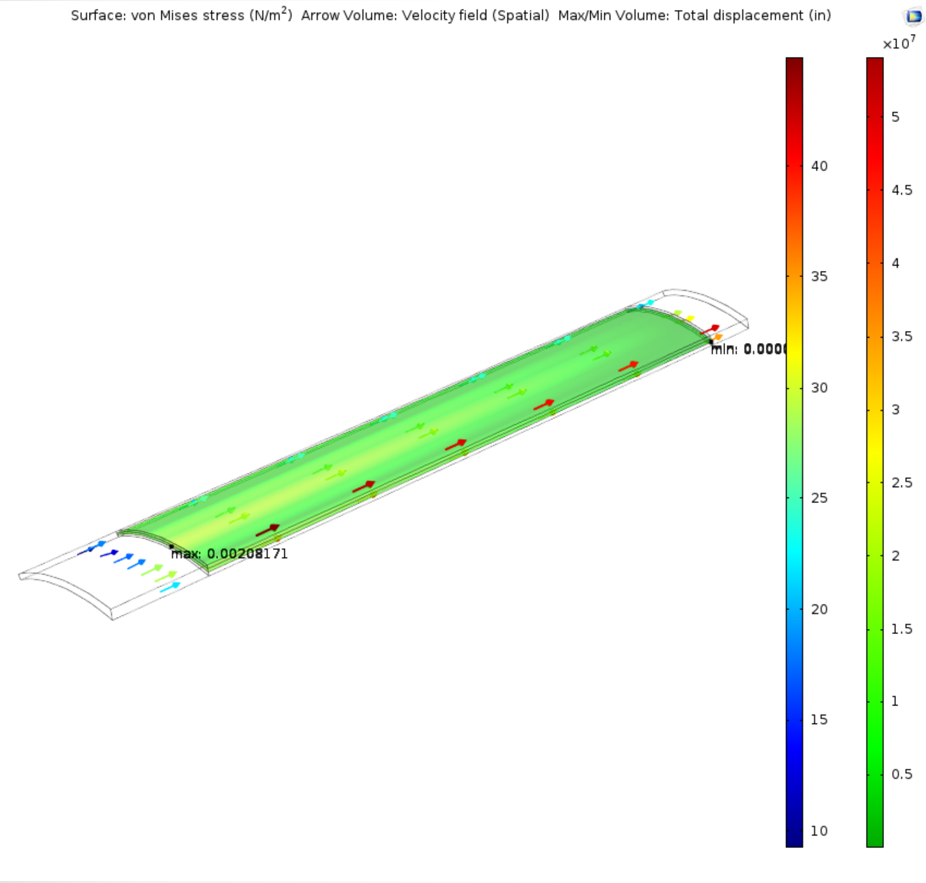
Segregated Solver

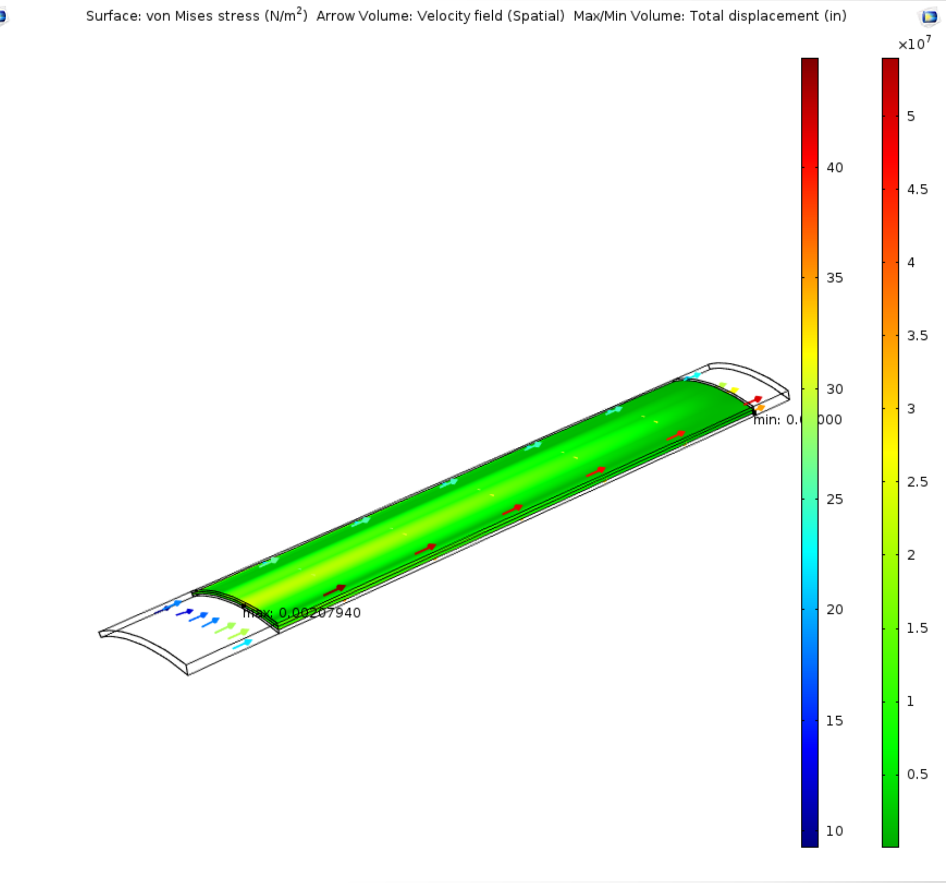
Fully Coupled Solver

- (Geometry, physics and Mesh Configurations are kept the same)
- Inlet velocity = 19 m/s (Max. Velocity ≈ 38m/s)

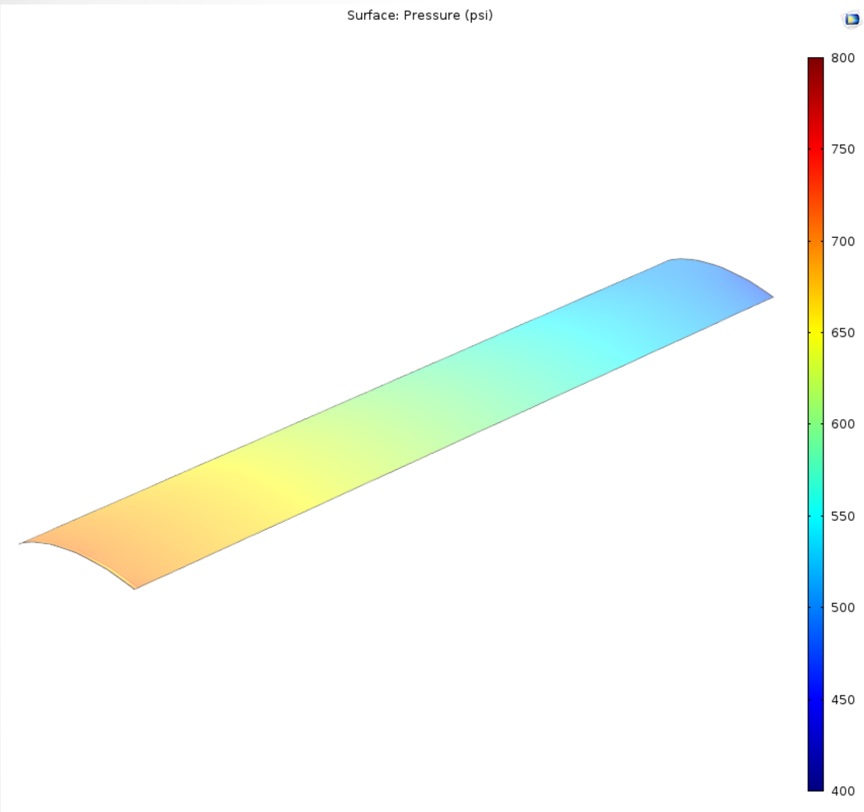
	Dof	Run Time	Hardware
Segregated	9,495,045	1 day 7 hours 32 minutes	24-core Intel-Xeon X-5650 @2.67GHz, 128 GB ram
Fully Coupled	9,495,045	2 days 1 hour 44 minutes	24-core Intel-Xeon E5-2695 v2 @ 2.40GHz, 256 GB ram

Stresses and max. displacement for the aluminum plate from segregated solution (left), and fully coupled solution (right)





Pressure over the upper surface of the aluminum plate based on segregated solution (left) and fully coupled solution (right)



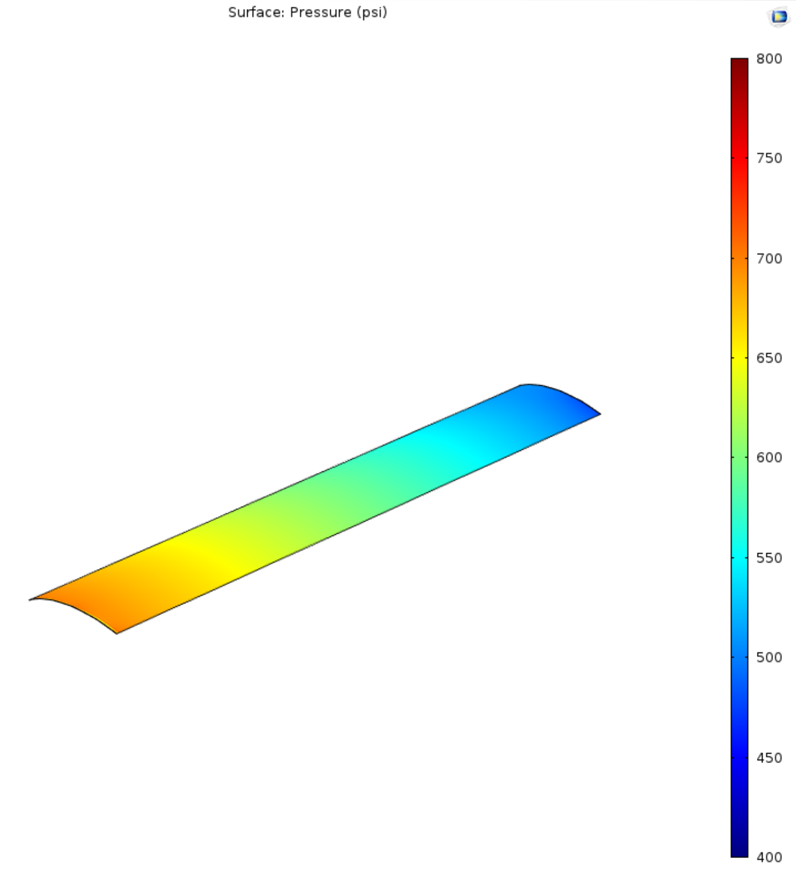
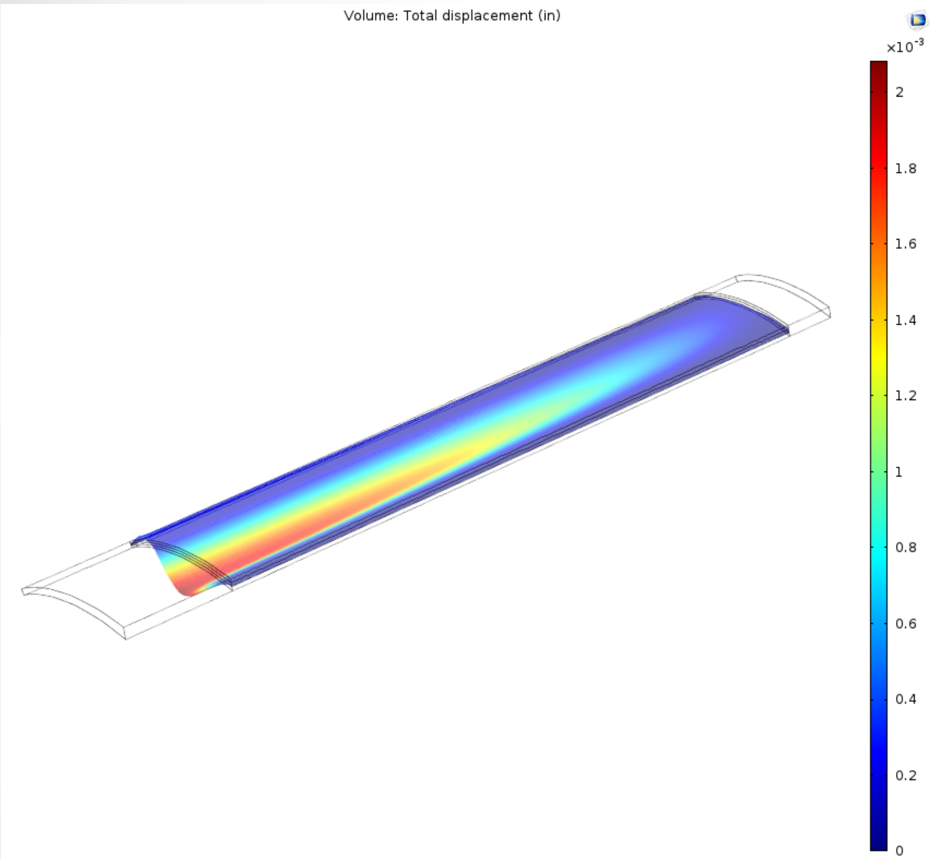
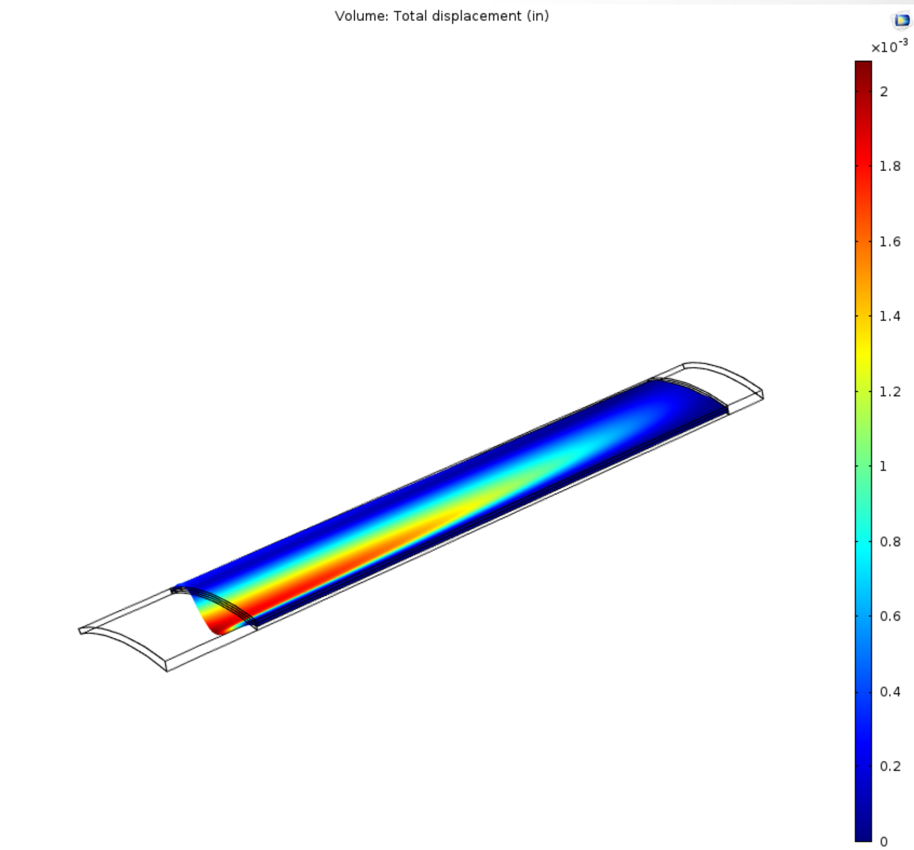


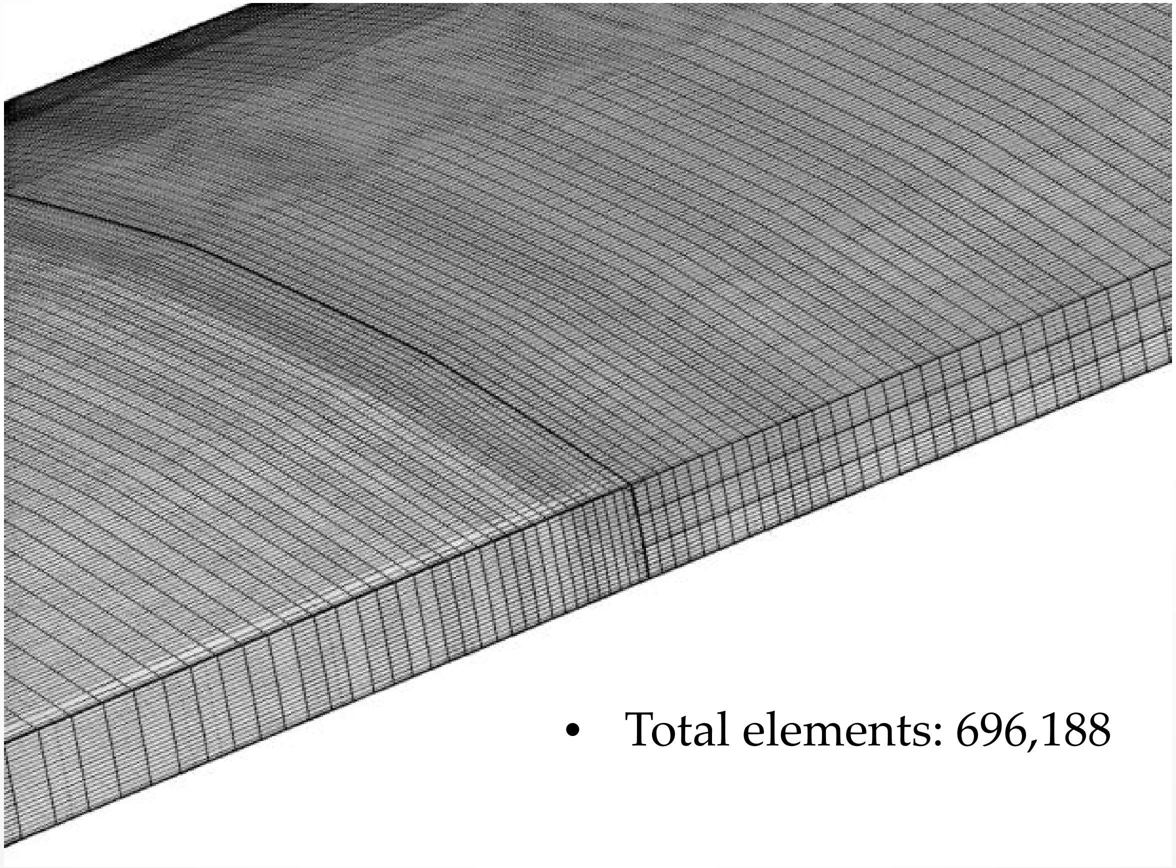
Plate deformation based on segregated solution on the left and fully coupled solution on the right





	Max. Displacement ($x10^{-3}$ in)	Pressure Drop (psi)
One-Way Coupling	1.93987	≈ 200 psi
Segregated	2.08171	≈ 200 psi
Fully Coupled	2.07940	≈ 200 psi

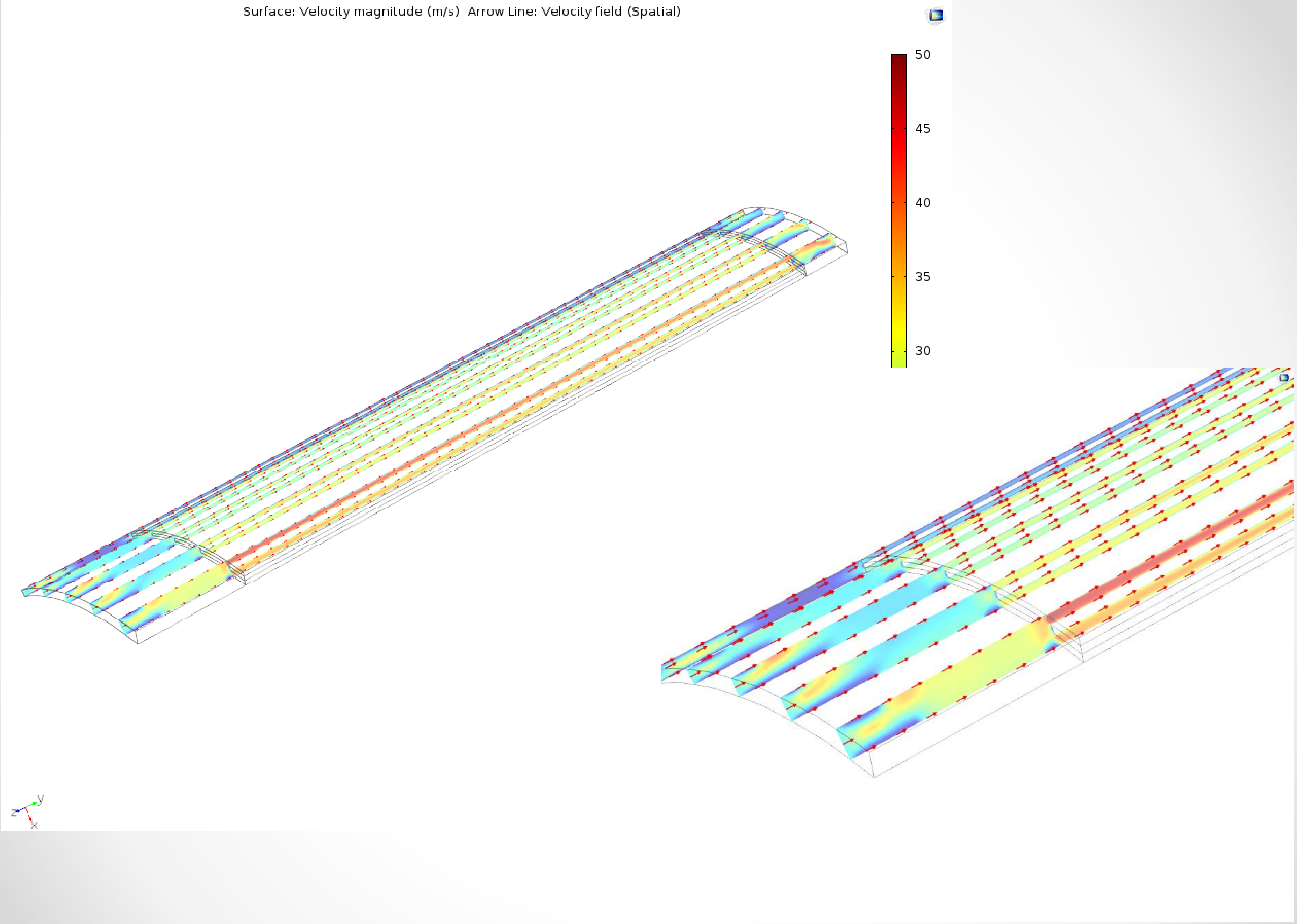
Mesh Quality



- Total elements: 696,188

Degree of freedom by variables

		Dof
Fluid	Velocity field	1,908,942
	Pressure	636,314
	Turbulent Kinetic Energy	636,314
	Turbulent dissipation rate	636,314
Moving Mesh	Spatial coordinates	1,908,942
Solid	Displacement field	3,768,219
	Total	9,495,045

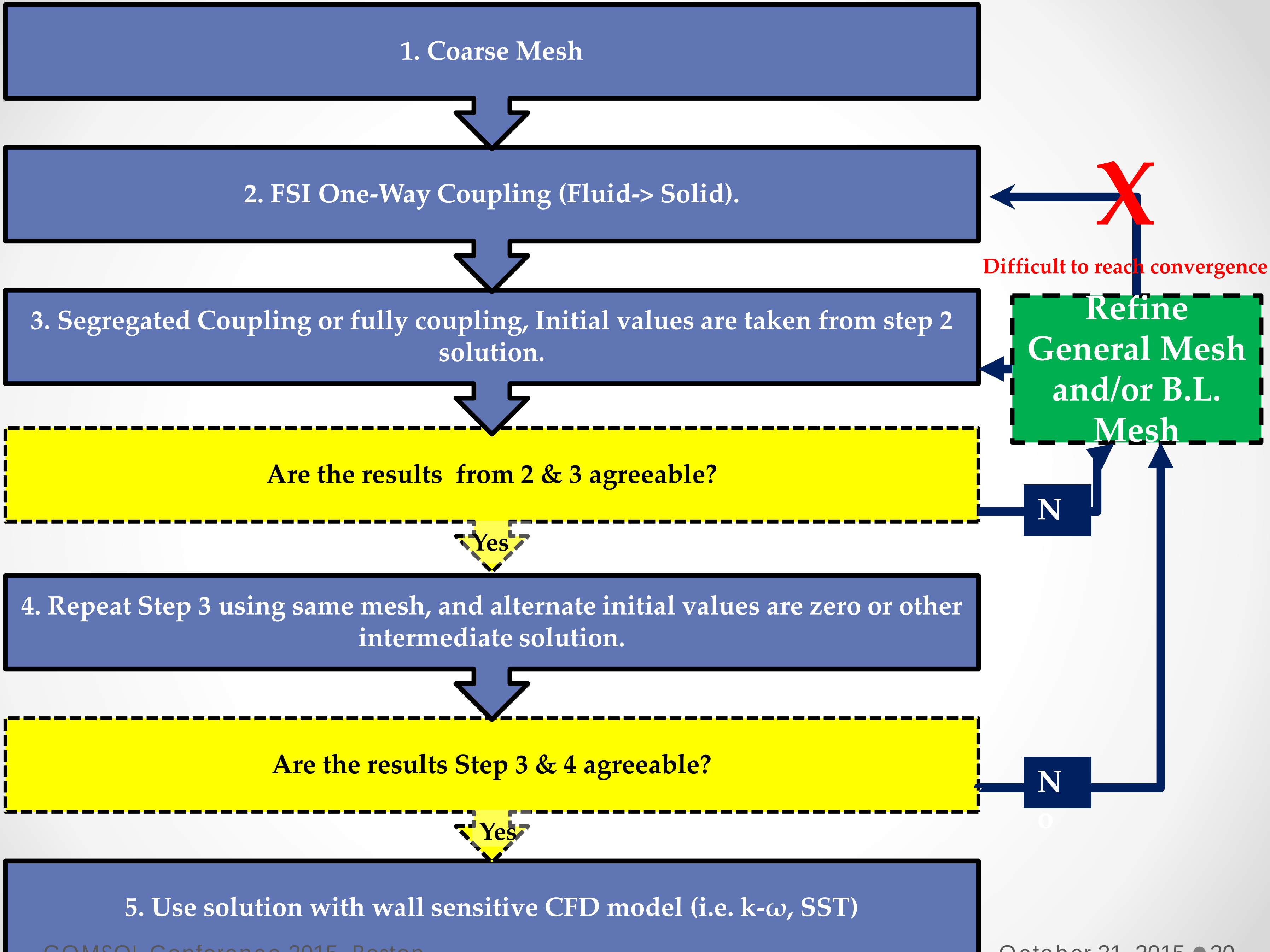


Surface: Velocity magnitude (m/s) Arrow Line: Velocity field (Spatial)
50
45
40
35
30

Step-2: Multi-Step solution approach

- The use of Segregated solver is now permissible
- However, convergence is difficult all physics is solved simultaneously from zero initial conditions.
- A multi-step approach is developed to achieve:

1. Optimization of computing time and capacity, while achieving a solution using high mesh resolution.
2. increase the likelihood to reach a converged solution.
3. Increase the reliability of the solution


1. Coarse Mesh
2. FSI One-Way Coupling (Fluid-> Solid).
X
Difficult to reach convergence
3. Segregated Coupling or fully coupling, Initial values are taken from step 2 solution.
Refine General Mesh and/or B.L. Mesh
Are the results from 2 & 3 agreeable?
N
Yes
4. Repeat Step 3 using same mesh, and alternate initial values are zero or other intermediate solution.
Are the results Step 3 & 4 agreeable?
N
o
Yes
5. Use solution with wall sensitive CFD model (i.e. k-ω, SST)

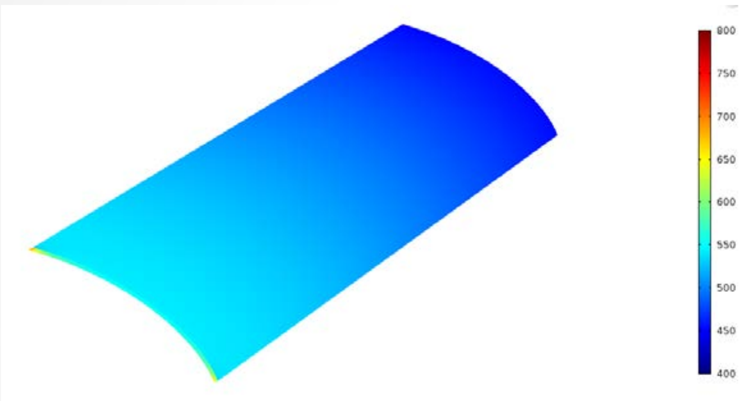
Step-3: Application of multistep approach on flow operating conditions

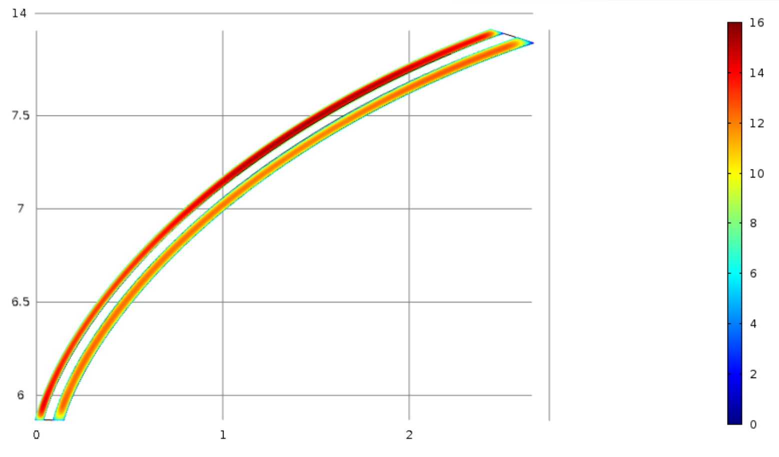
- Solving Based on nominal and extreme channel dimensions

Flow at the inlet is 8 m/s for all cases

Case #	Designated name	Description
1	U44-L56	0.0044 inch-thick flow channel on the convex side and 0.0056 inch-thick on the concave side
2	U56-L44	0.0056 inch-thick flow channel on the convex side and 0.0044 inch-thick on the concave side
3	U50-L50	0.0050 inch-thick flow channel on both sides(Nominal dimension of the flow channel)

Results

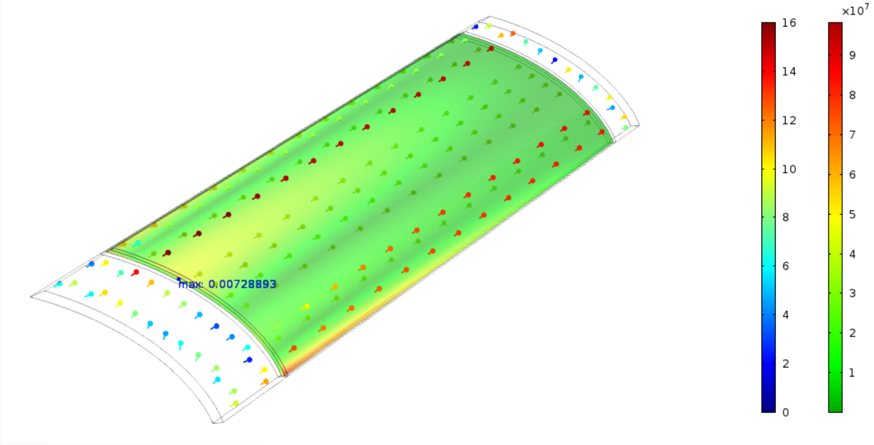


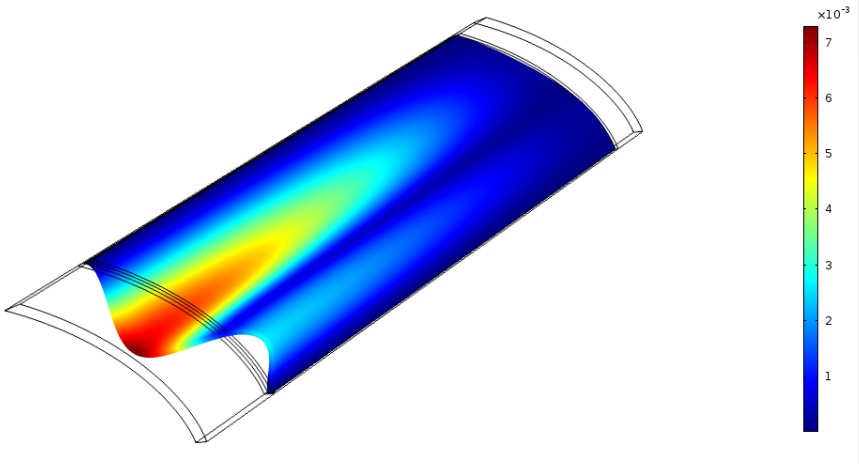


The pressure drop across the channel is about 100 psi which in agreement with true value of recorded at HFIR.

Cross section velocity contour at mid-plane indicate faster flow profile on the narrower side (convex side).

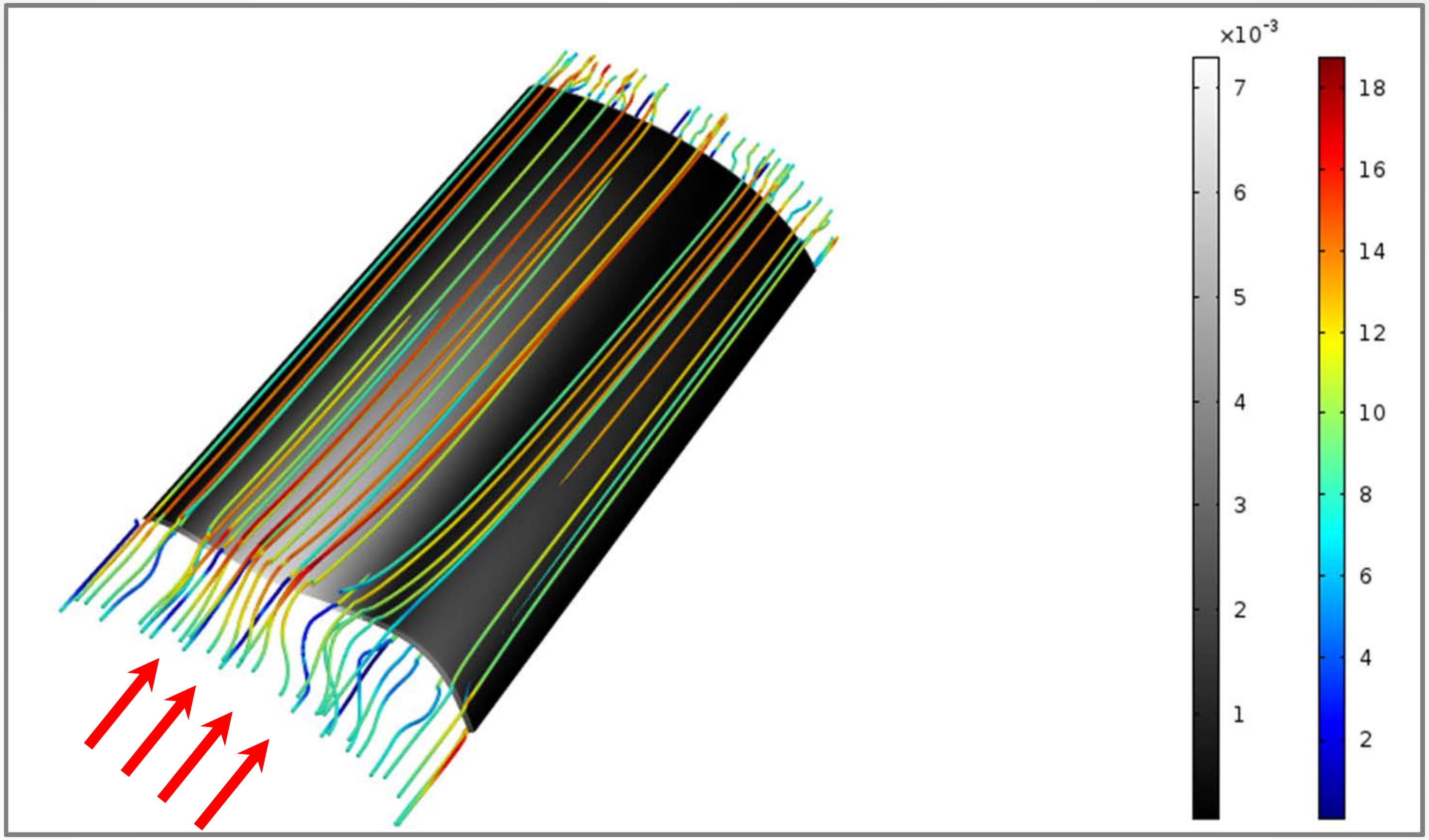
Results





The colored surface indicate the stress distribution over the plate surface, colored arrows indicate the velocity magnitude and direction, and the location and maximum deflection is marked on the surface

7-mil maximum deformation occurs close to the leading edge, with S-type deformation



Flow streamlines over fuel plate deformed surface

for steady-state solution.

Summary and conclusion

- Using segregated and fully coupled solvers produced similar results.
- This allowed the use of “the less costly” segregated solver.
- A multi-step methodology is developed to improve the convergence rate and to produce reliable results.
- Applying this multi-step approach on cases with nominal and extreme channel dimensions

Future Work

- The results of this analysis will directly support ongoing best-estimate design and safety analysis by (Research Reactors Division) RRD which include nuclear and thermal analysis. Next Step is to use the FSI model developed here and combine it with the thermal analysis currently under development by Dr. James Freels.

Acknowledgements

Dr. James Freels at ORNL

-This work was supported in part by the U.S. Department of Energy, Office of Science, Office of Workforce Development for Teachers and Scientists (WDTS) under the Visiting Faculty Program (VFP).