Validation of Turbulent Mixing in the NNS Chimney

Breken Wallar, CORE Nuclear Engineering Intern

NIST Center for Neutron Research
100 Bureau Dr., 20899 Gaithersburg, MD, USA

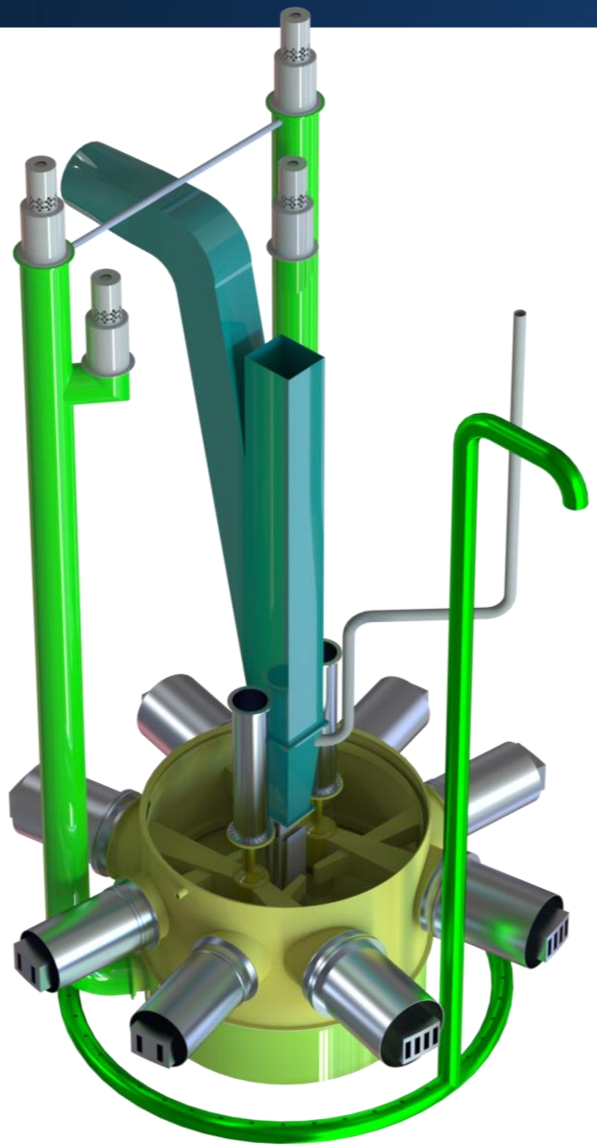
About Me

21-Year-old incoming senior at Texas A&M's department of Nuclear Engineering

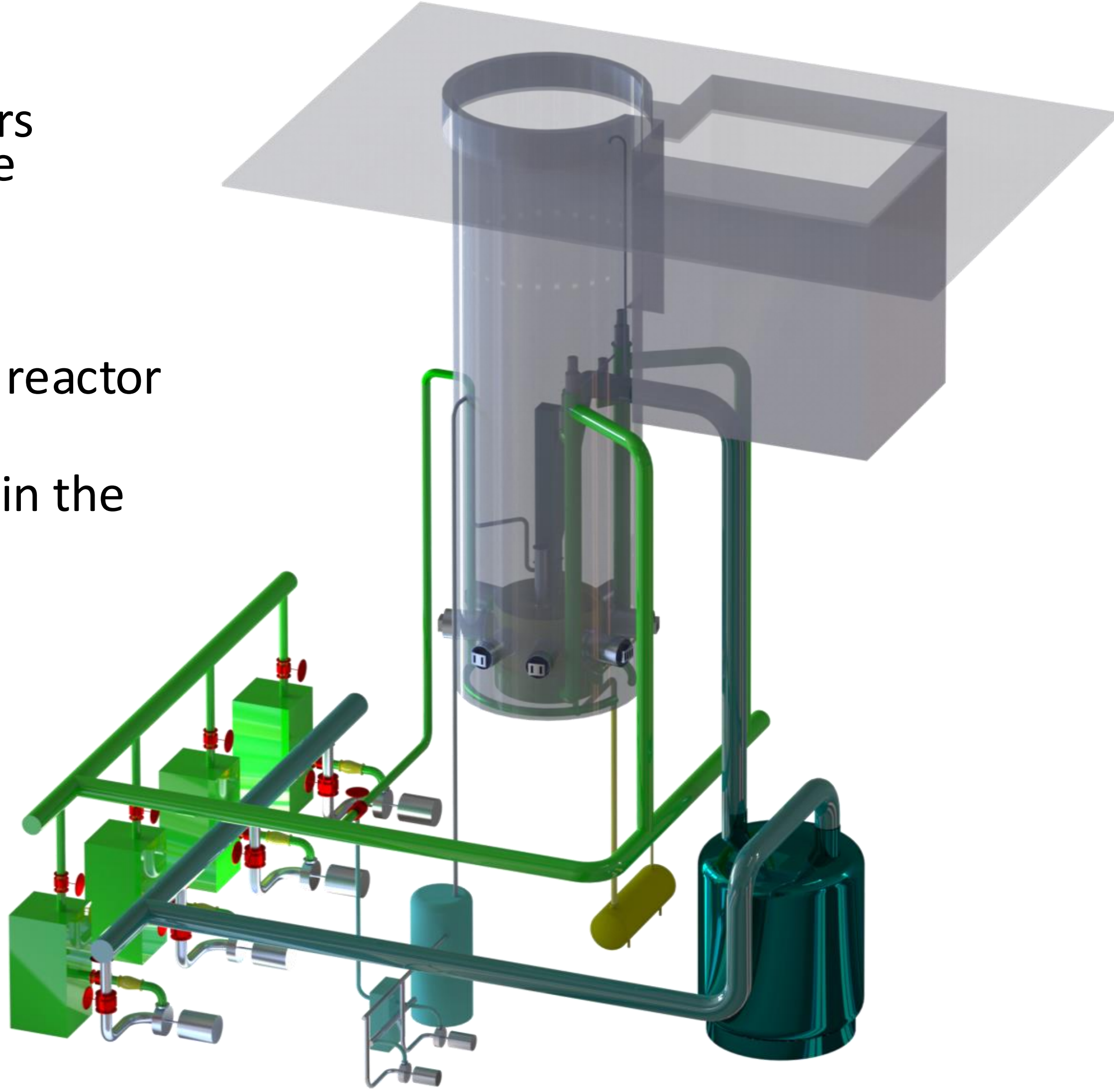
I love understanding the world around me, and numerical modeling provides a way to understand a specific geometry REALLY well

I also love my girlfriend Emma

Design of NNS Overview



- Influenced by several reactors designed for neutron science
- Nominal power of 20 MW
- U-10Mo LEU (or U3Si2)
- Light-water-cooled compact reactor core
- Surrounded by heavy-water in the reflector tank
- 2 Cold Neutron Sources
- 8 Thermal Neutron Beams
- 40 days operating cycle



Reflector tank with surrounding features

Reactor Pool and Primary Coolant System

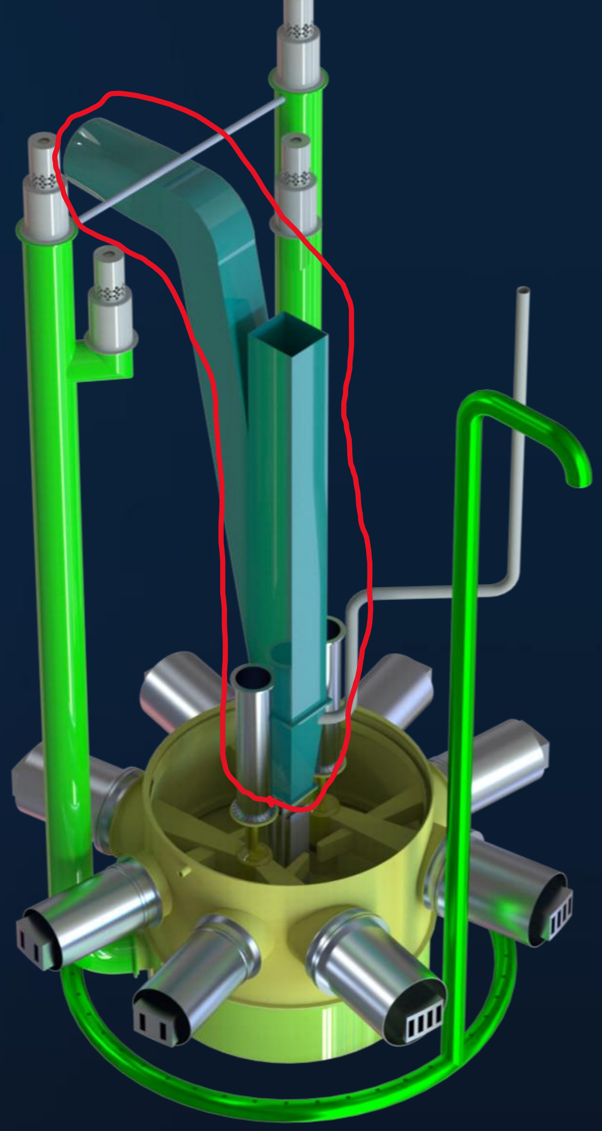
The Project Scope

- RADIOACTIVE WASTE MIGHT SPEW INTO THE NNS POOL

- To ensure this does not happen we need to know what is happening in the chimney

- To perform this numerical modeling, a commercial CFD code, ANSYS FLUENT, will be utilized

- Lower resolution system codes have already been used, higher Resolution modeling is the next step



CFD Basics

1. Decide on a domain of interest for the problem i.e., NNS chimney/pool

2. Discretize the domain into lots of small control volumes i.e., making a mesh

3. Develop a numerical model i.e., adjusting the setup of ANSYS FLUENT

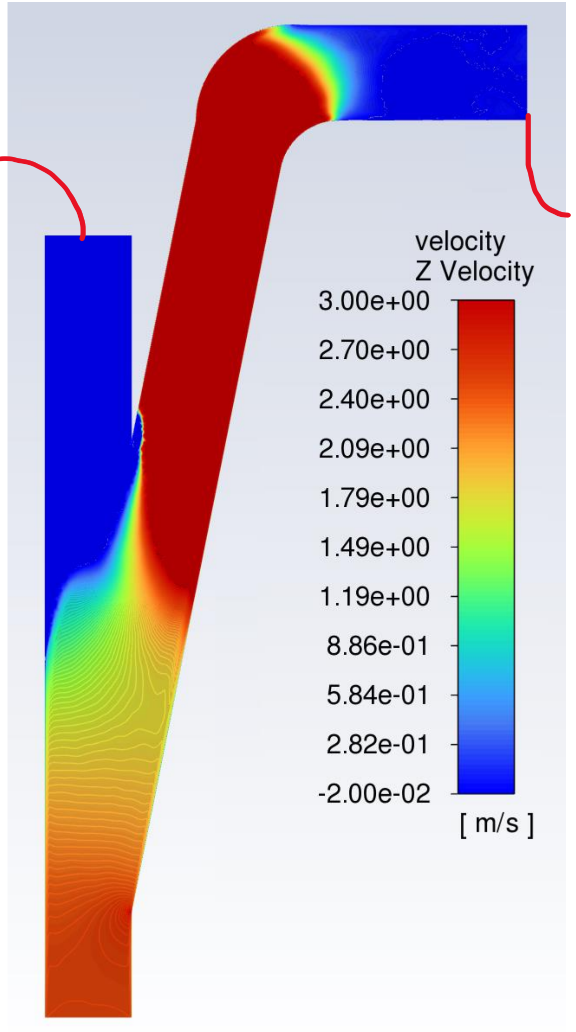
4. Process the data to develop conclusions about results i.e., look at pretty pictures

5. Repeat until a model with acceptable accuracy is created

Model of NNS Chimney Control Volume

Chimney Pool Interface

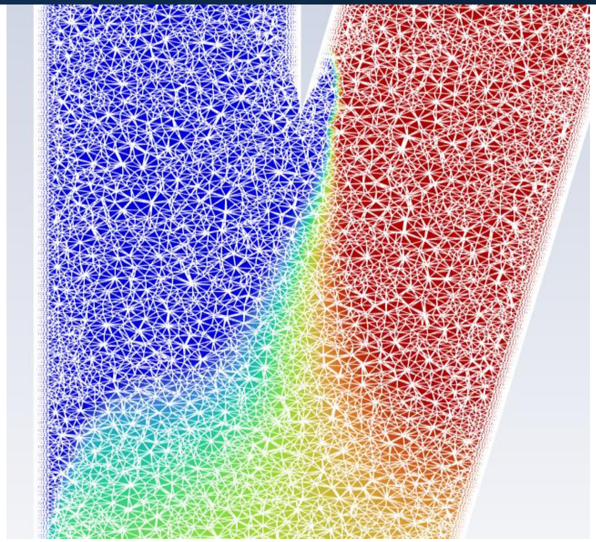
50 kPa-gage



Outlet

0 gage pressure

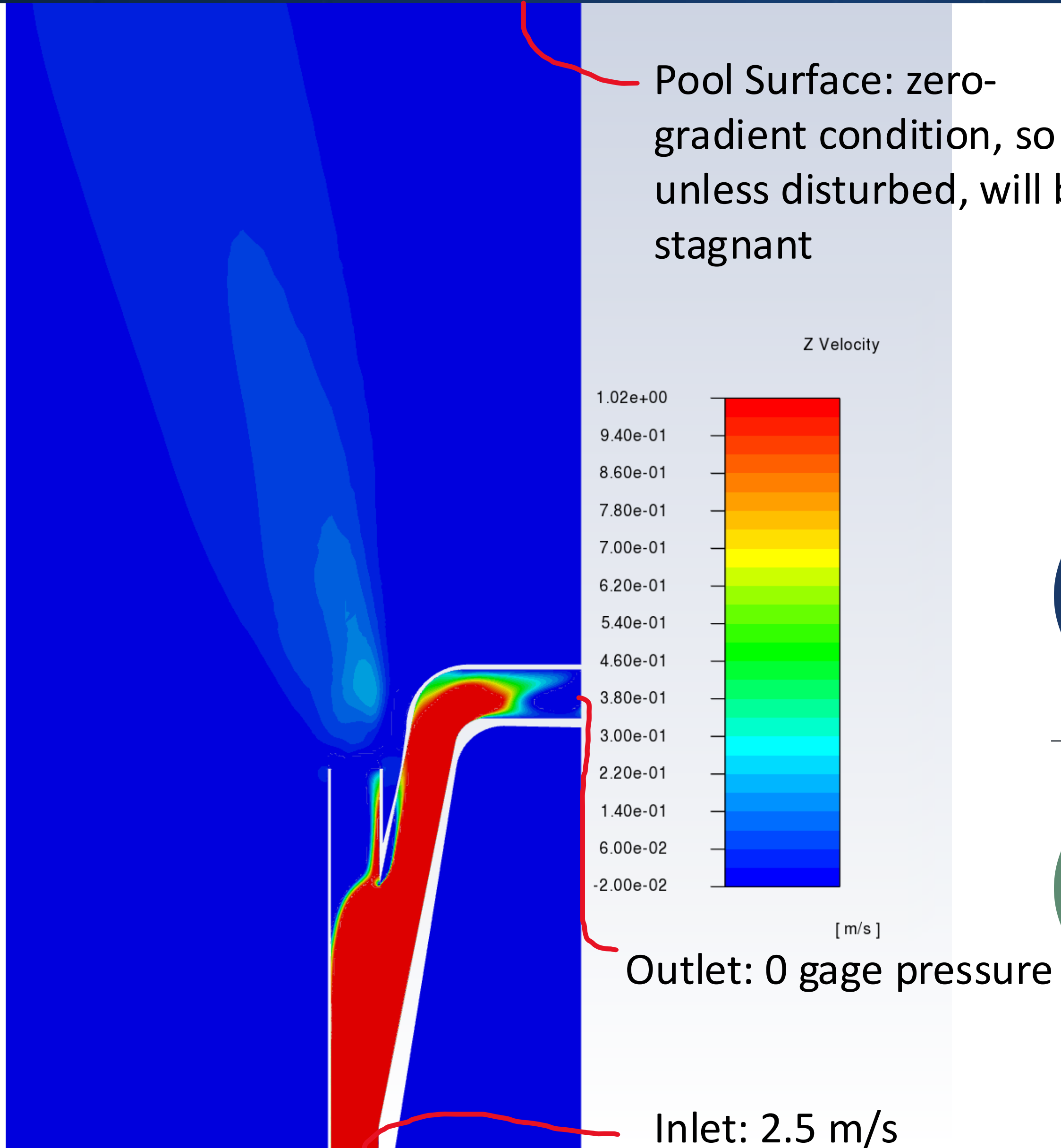
Inlet: 2.5 m/s

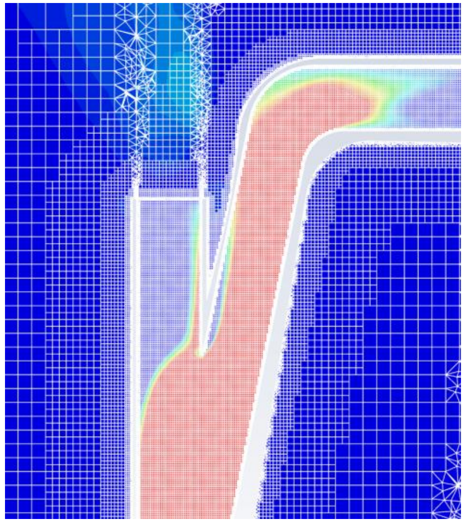


The NNS Chimney was discretized into 1.8 million tetrahedral (pyramid shaped) cells

Large amounts of backflow from the pool pushes the incoming flow into the hot leg entirely leaving no backflow through the pool-chimney interface.

NNS Chimney With Pool





The pool and chimney are simulated with 3.7 million hex cells

The incoming flow is not entirely redirected resulting in a plume of radioactive particles shooting into the NNS pool.

Verification and Validation

The two modeling approaches give different results. To clear up this disagreement a verification and validation procedure can be carried out.

Verification refers to the ability of a code to run mathematical computations mistake free. ANSYS FLUENT is already a verified CFD code.

Validation is the process of determining the accuracy of a model to some experimental data of interest. This will be the central goal, until complete, further progress on NNS modeling cannot be reliably performed.

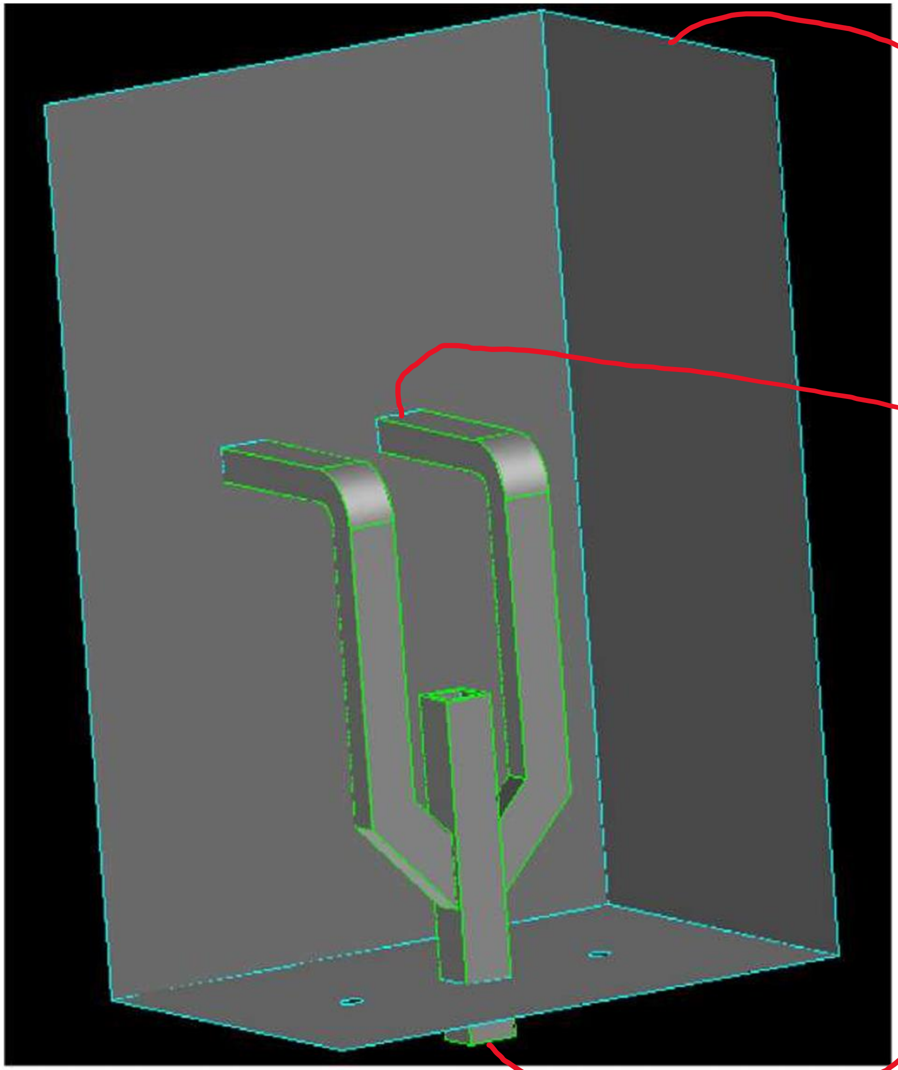
Validation Subject: Sengupta et al.

This video shows a scale model of a pool type reactor chimney during operation. The flow is injected with dye to help visualize how the fission product concentration develops.

This is almost identical to the NNS scenario. The main difference is in the number of outlets.

The experimental data will be used to validate numerical models developed for the NNS

Domain and Boundary conditions



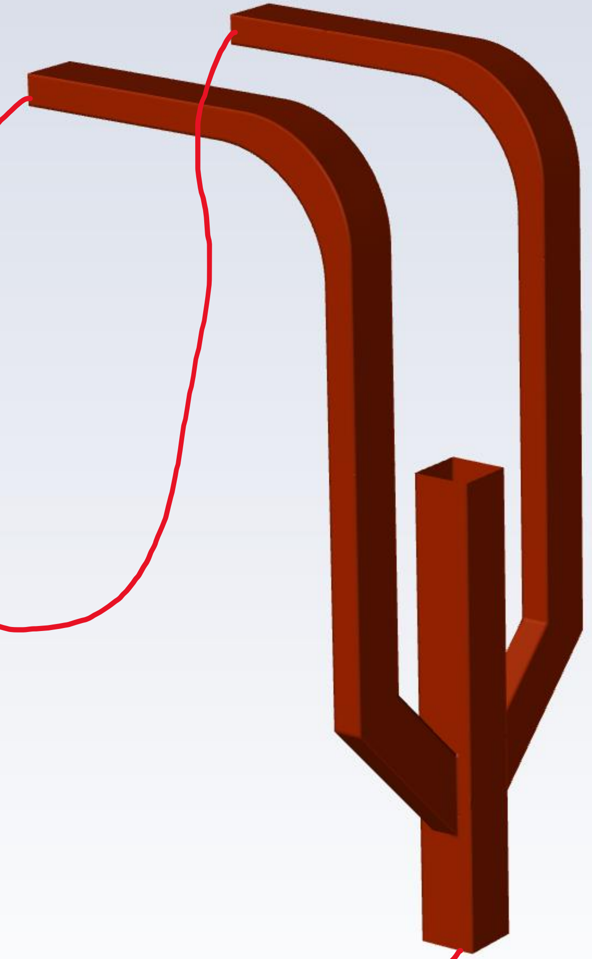
Pool Surface:
zero-Gradient

Outlet:
Specified
Pressure

Inlet: 0.5 kg/s

(a) Computational domain Sengupta et al.

<u>ALL WALLS ARE NO-SLIP</u>



Pool Surface:
Zero-Gradient

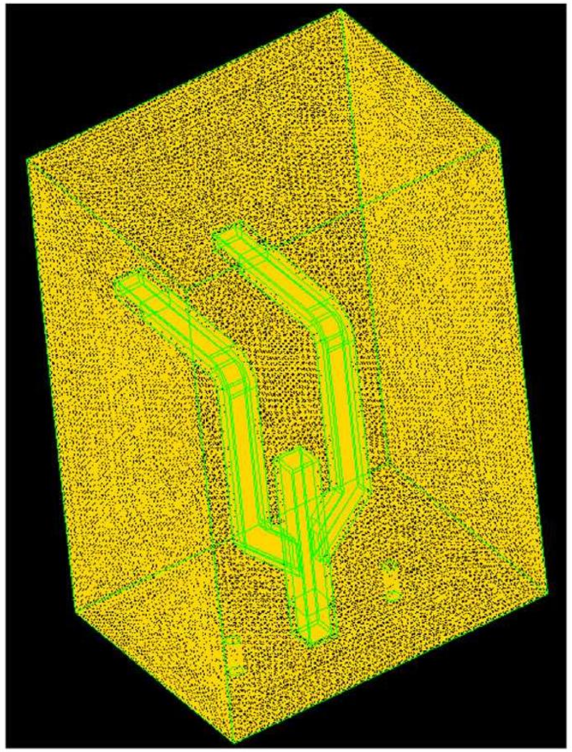
Outlet 1:
50% outflow

Outlet 2:
50% outflow

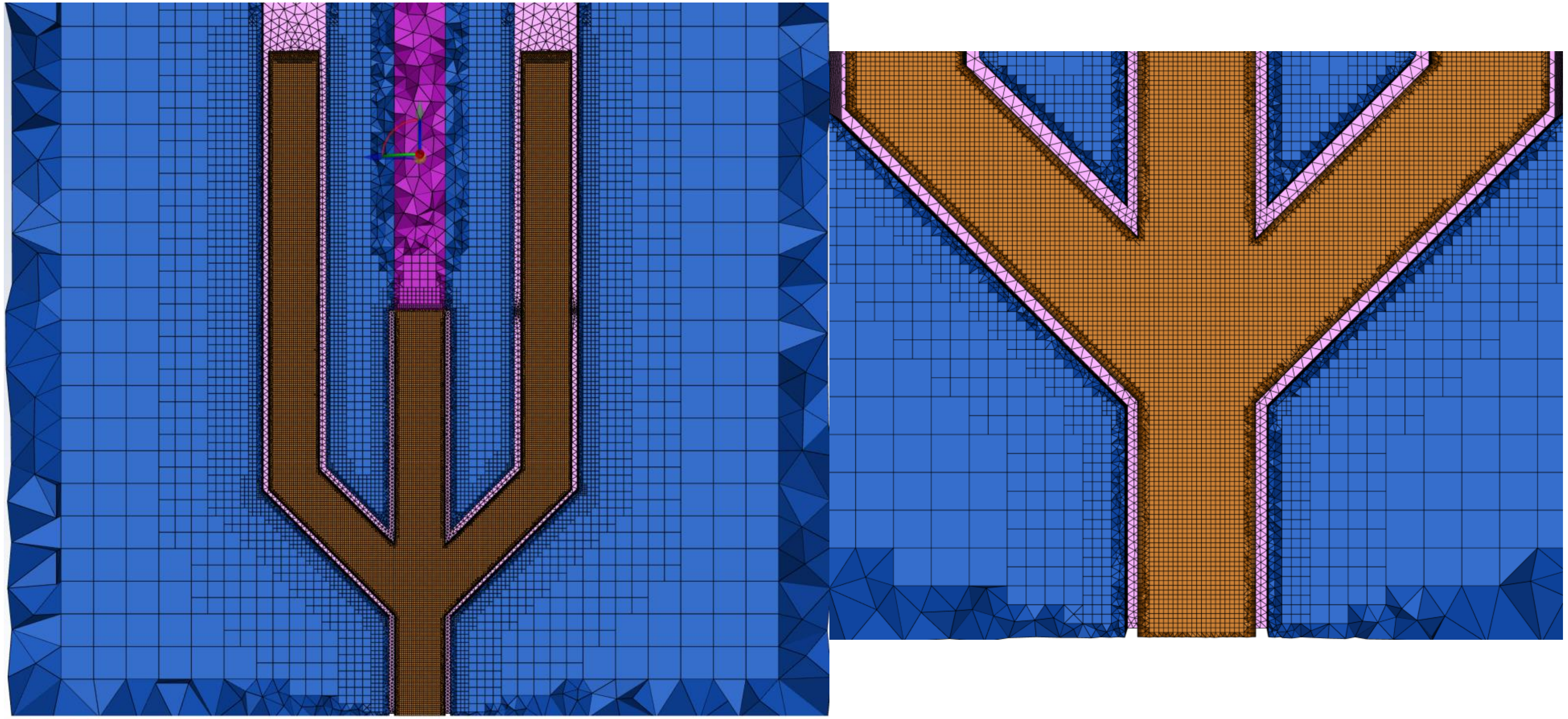
Inlet: 0.5 kg/s

Present Domain without pool shown

Discretization



(b) Mesh used in computation



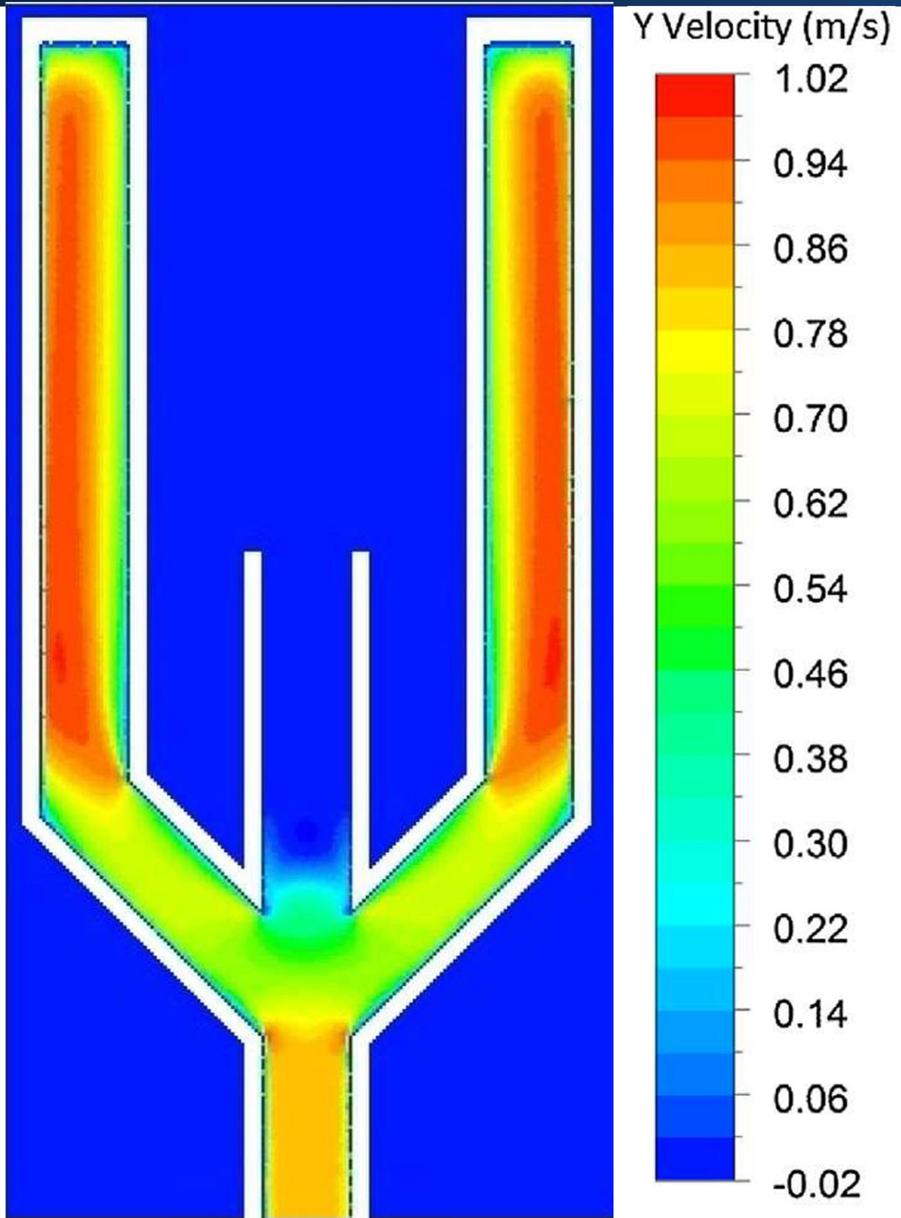
Present Mesh

Numerical Model Setup

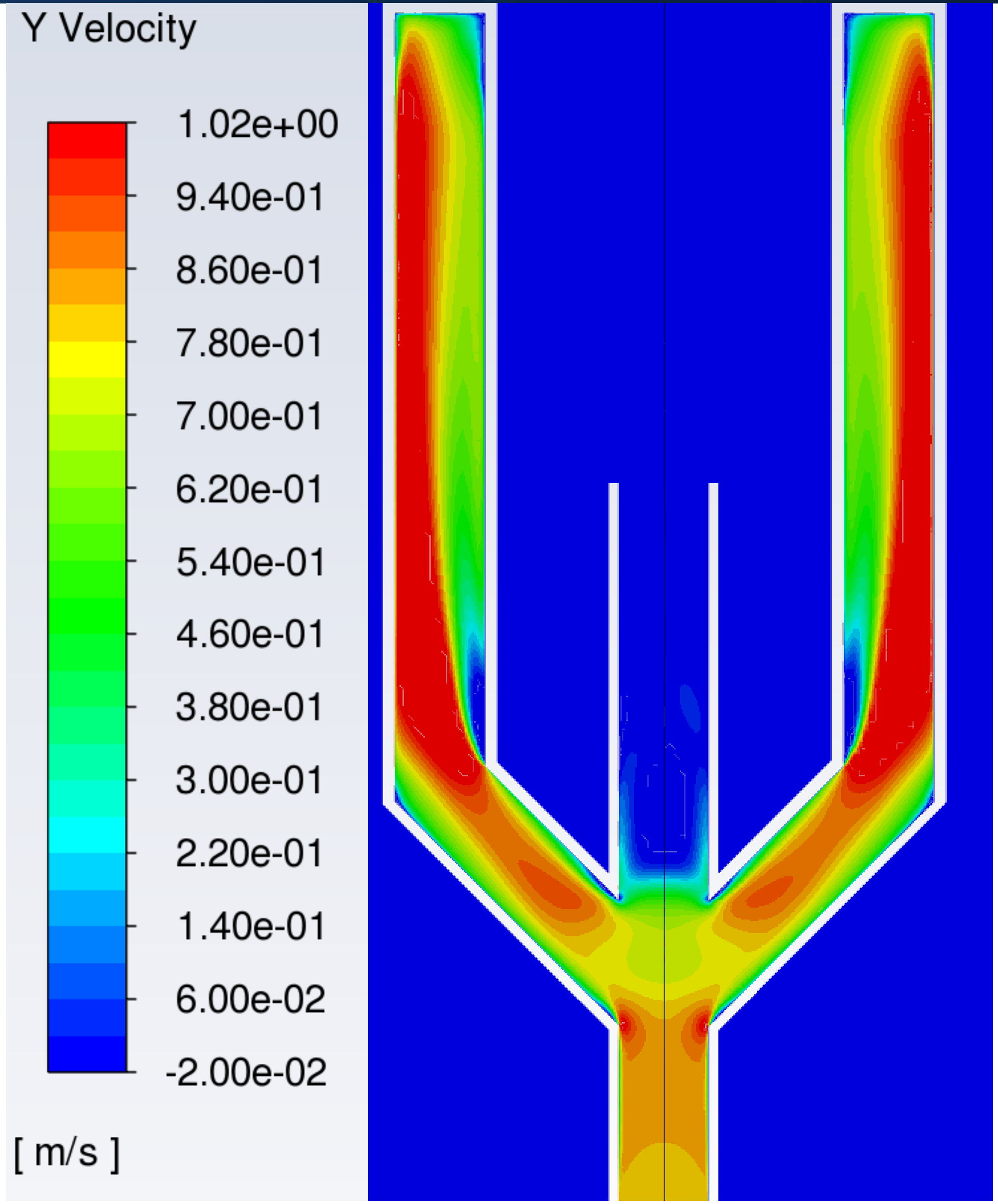
Similarities and differences between Sengupta et al. and the present work are listed in the table below.

Similarities	Differences (present vs Sengupta)
Turbulence model: K-omega SST	Schemes: Coupled vs SIMPLEC
Inlet condition: 0.5 kg/s	Outlet Condition: Outflow vs specified pressure
Steady State Simulations	FLUENT VERSION: 2022 R2 vs $\leq$2014

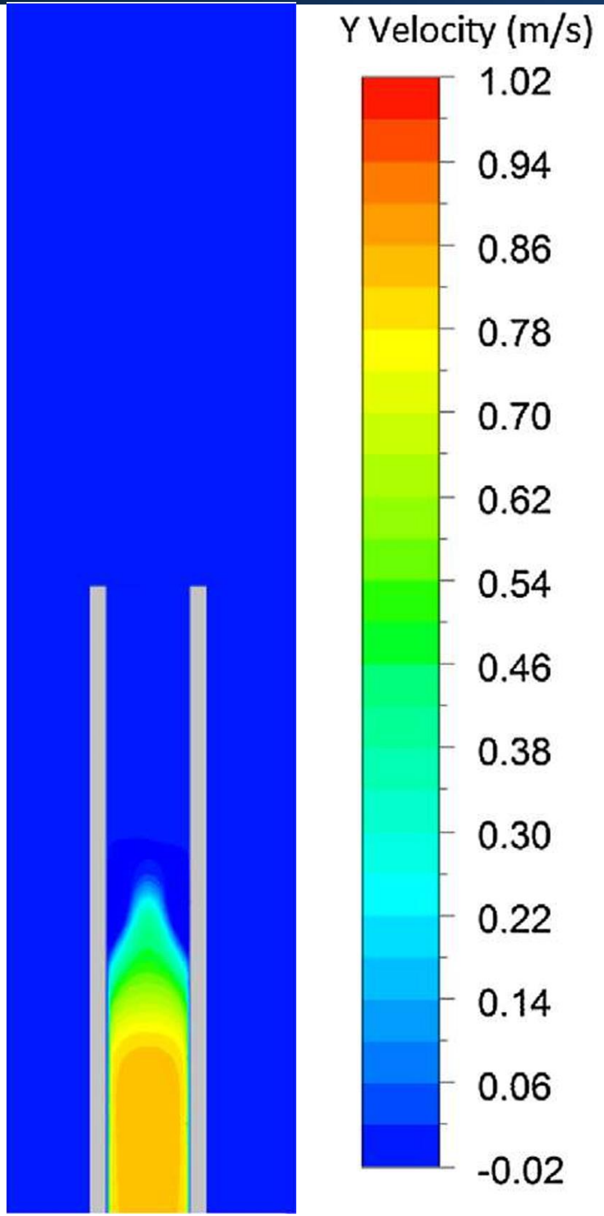
Numerical Velocity Contour Comparison

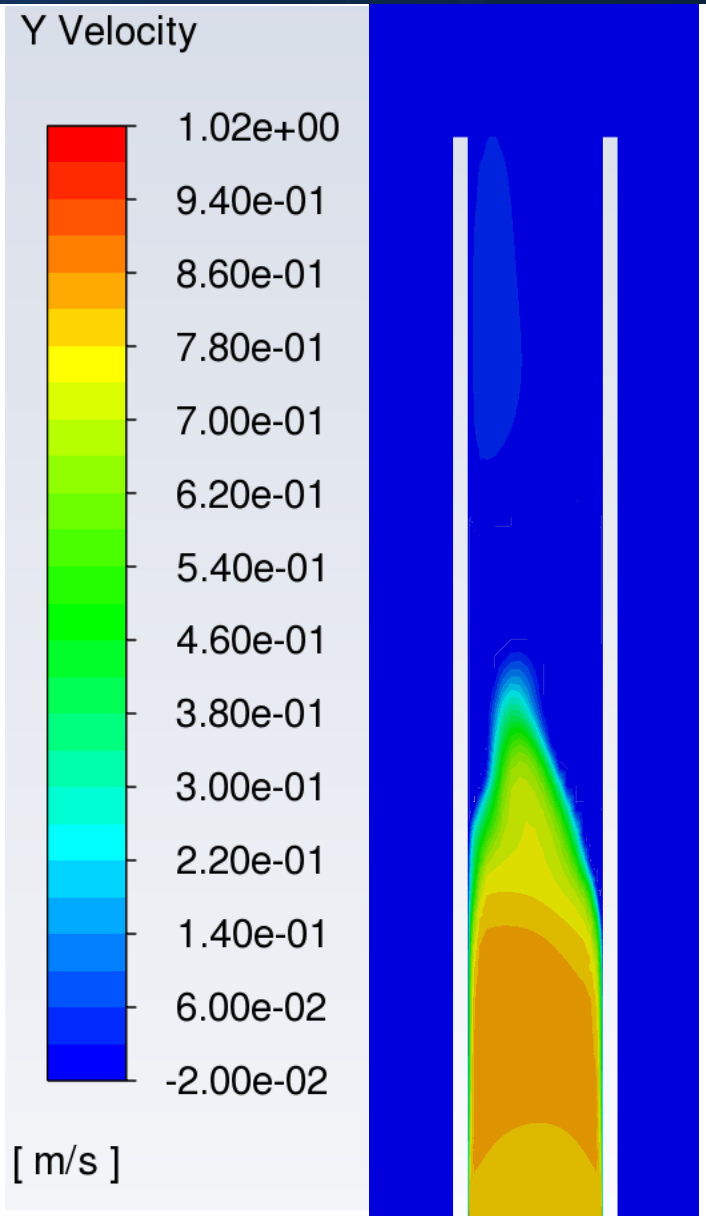


X-Y Plane

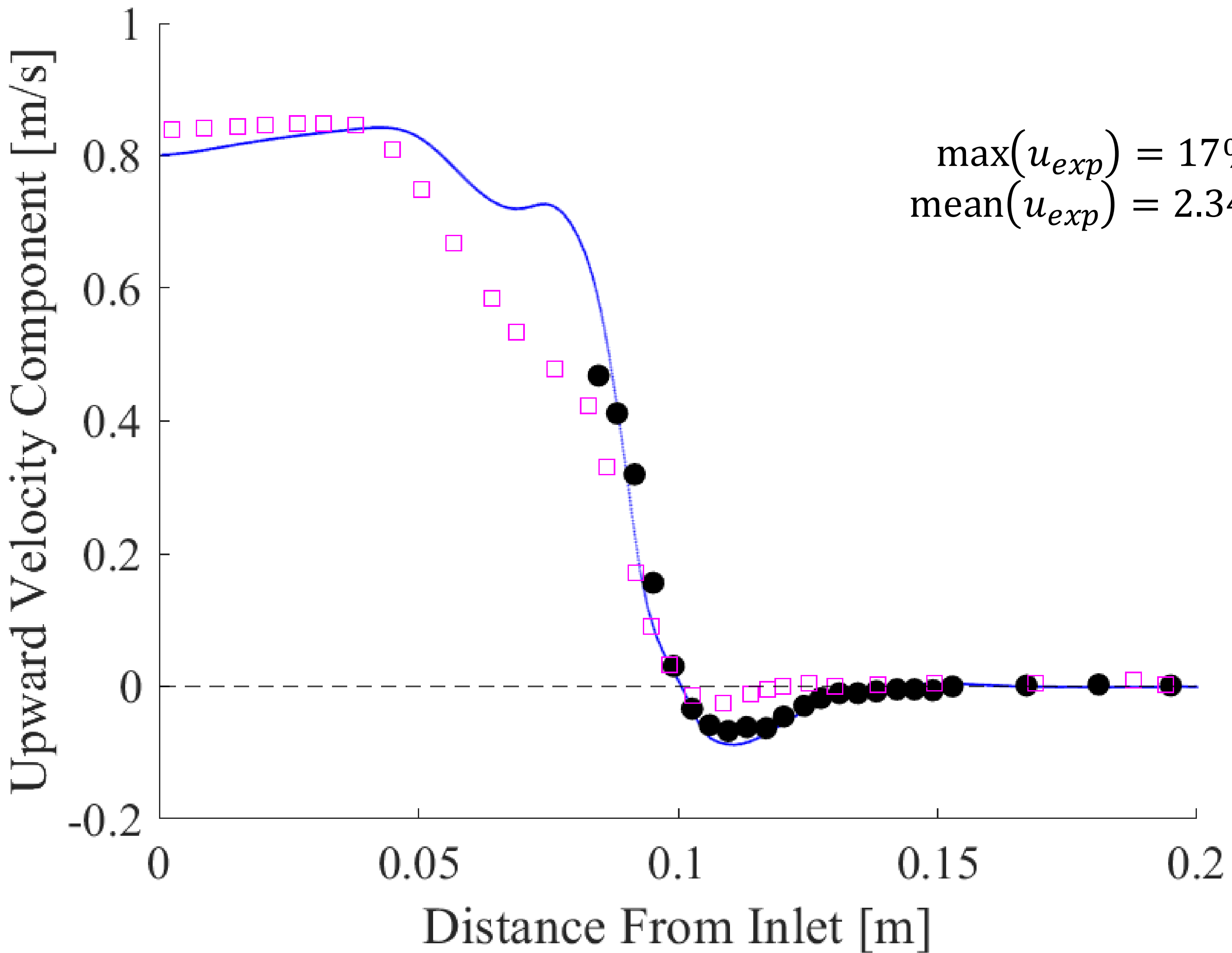


Y-Z Plane Velocity Contours



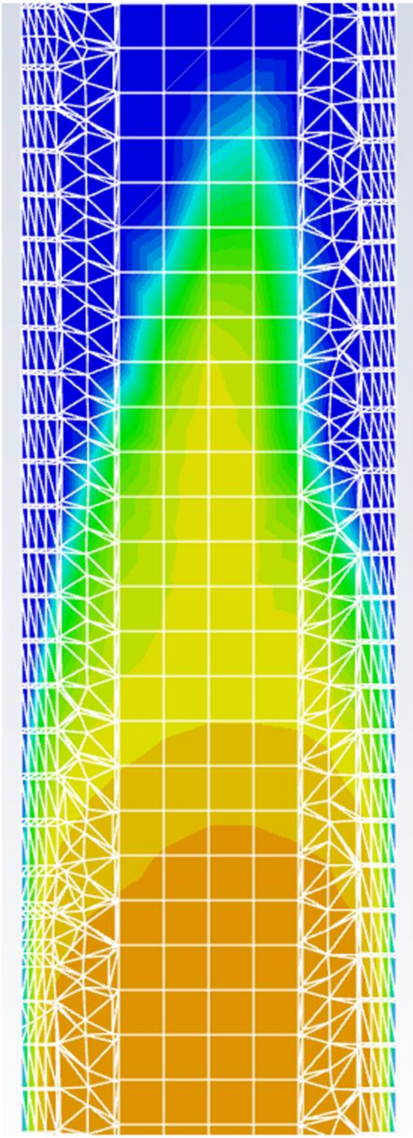


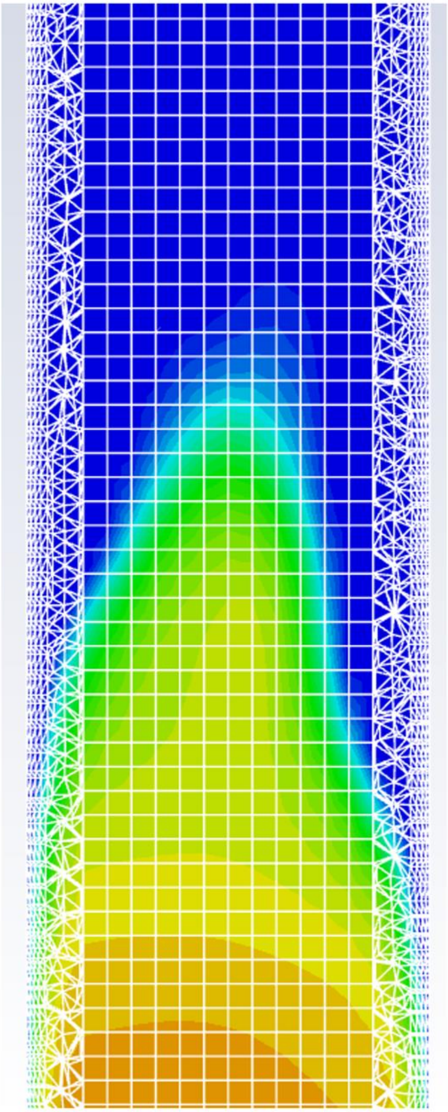
Quantitative Results

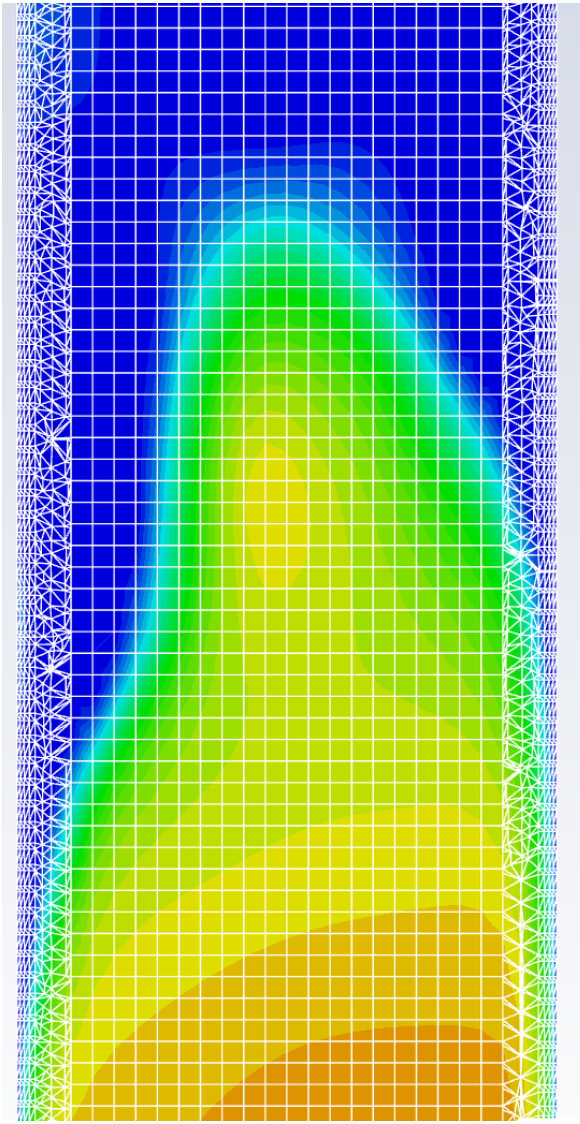


$$\max(u_{exp}) = 17\% \, [U_{inlet}]$$
$$\mathrm{mean}(u_{exp}) = 2.34\% \, [U_{inlet}]$$

Mesh Convergence


Coarse @ 50k iterations

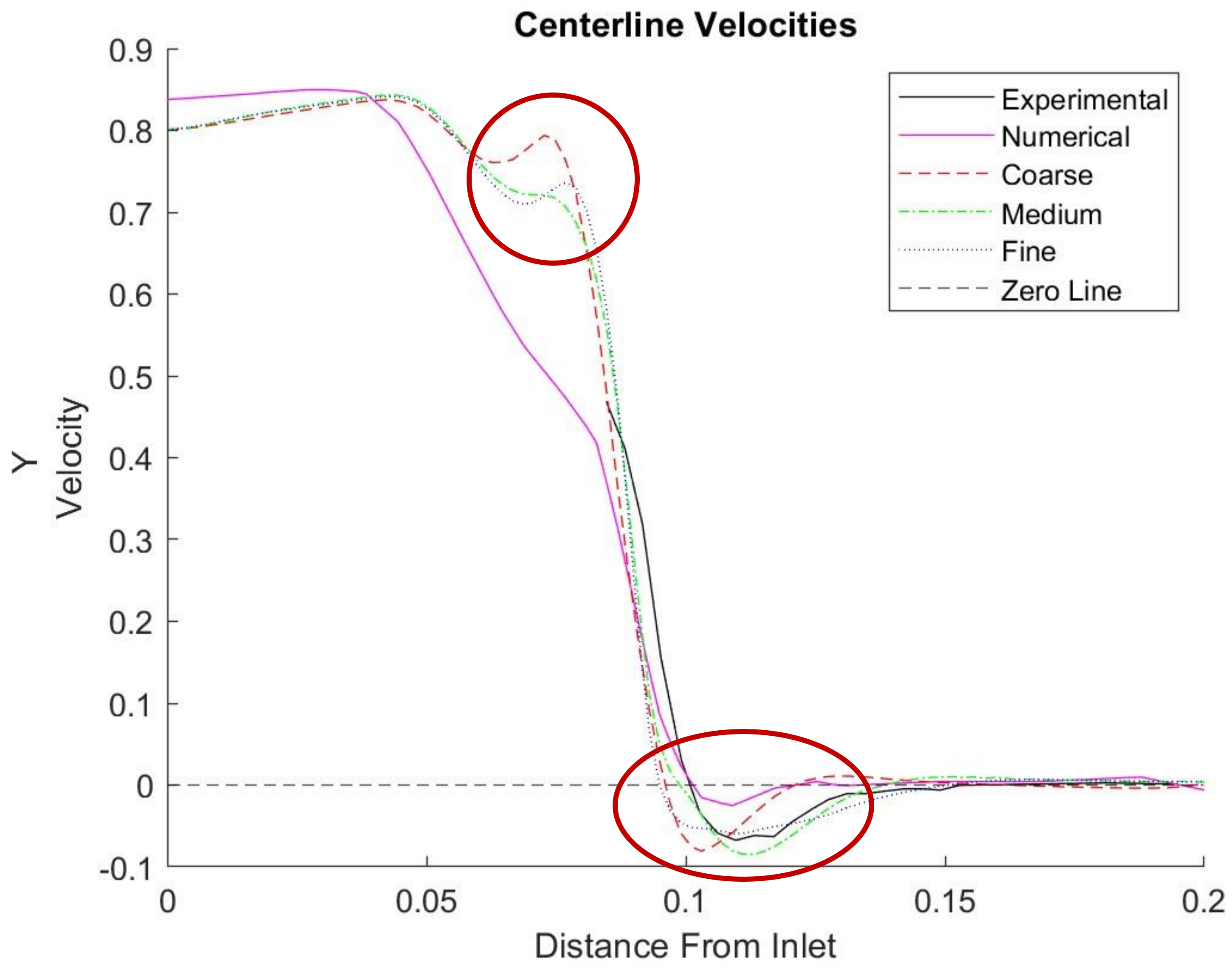

Medium @ 28k iterations


Fine @ 14k iterations

The previous slide showed how the medium mesh compared to experiment.

This slide shows qualitatively the difference in results given different resolutions of mesh

Quantitative Comparison



10 separate iterations taken 5 iterations apart and then averaged together for each mesh.

The red circles show the locations where the different meshes show the most discrepancy

Further work must be done to derive a Grid Convergence Index that describes the error introduced by discretization

Conclusions

NIST

ANSYS FLUENT 2022 is able to accurately (within 3%) model the upward velocity in the chimney of a pool type reactor with a 4-way junction

Experimental data is more closely aligned with present analyses than previous ones.

NNS models can now be developed with this error in mind, and design of the chimney can commence

References

[1] Sengupta, Samiran, et al. “Piv investigations on the turbulent mixing of two opposing flows inside a scaled chimney model of a research reactor.” *Experimental Thermal and Fluid Science*, vol. 63, May 2015, pp. 115–132, https://doi.org/10.1016/j.expthermflusci.2015.01.014.

Acknowledgements

Huge thank you to my mentor Abdullah Weiss and friend Evan Bures for making this process fun and manageable!

Wouldn't have wanted to do it as much without you guys

Another huge thank you to CHRNS for providing the funding for the CORE program and allowing this work to be carried out!

Questions??

Breken Wallar
NIST Center for Neutron Research
100 Bureau Drive, Gaithersburg, 20899, USA