A FLUENT CFD Model for the OSU Calorimeter for Rate of Heat Release Predictions

A Status Report

Garrett Cappello
Graduate Student

Bakhtier Farouk
Professor

Mechanical Engineering and Mechanics
Drexel University, Philadelphia PA

***Tenth Triennial International Aircraft Fire and Cabin Safety Research Conference*,**

October 17-20, 2022, Resorts Casino and Hotel, Atlantic City,, NJ

Outline of the presentation

- **Background and a bit of history**
- **Research areas and topics**
- **Progress to date, October 2022**
- **Accomplishments and concluding remarks**

The OSU Rate of Release Apparatus (ASTM E906)

A brief history

- **ASTM STP 502,** *Smith*, **1972**
- **Fire Safety Journal,** *Babrauskas*, **5, 1, 1982**
- **Fire and Materials,** *Tran*, **12, 1988**
- **Fire Safety Journal,** *Filipczak, Crowley and Lyon*, **40, 7. 2005**
- **DOT/FAA/TC-TN15/34,** *Lyon, Fulmer, Walters, and Crowley*, **2016**

Related work:

- **NIST Technical Note 2077,** Bryant and Bundy, **2019**

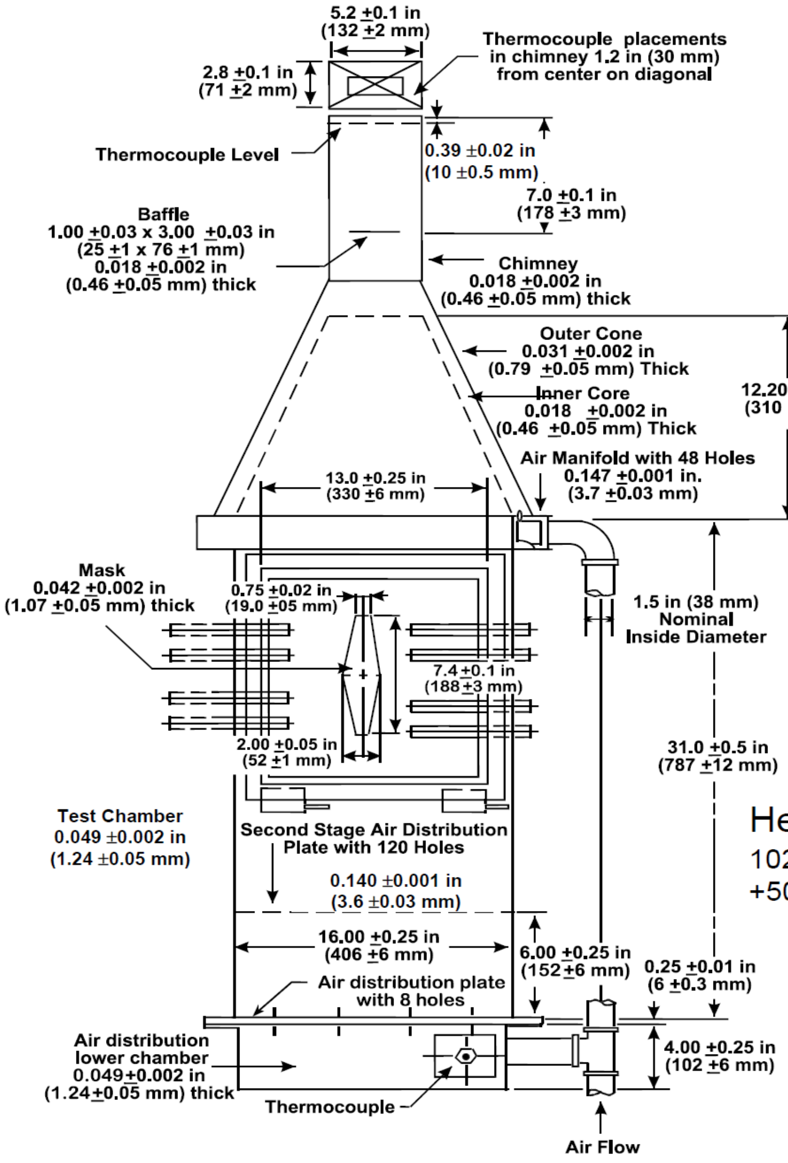
Title 14 Code of Federal Regulations (CFR) Part 25.853. with a calibrated radiant heat flux of 35 kW/m^2

- **Poor agreement between OSU fire calorimeter HR and HRR results obtained in different laboratories.**

- **A study focused on whether *differences in the airflow* through three "Title 14 CFR 25.853" fire calorimeters could account for the poor reproducibility of the HR and HRR.**

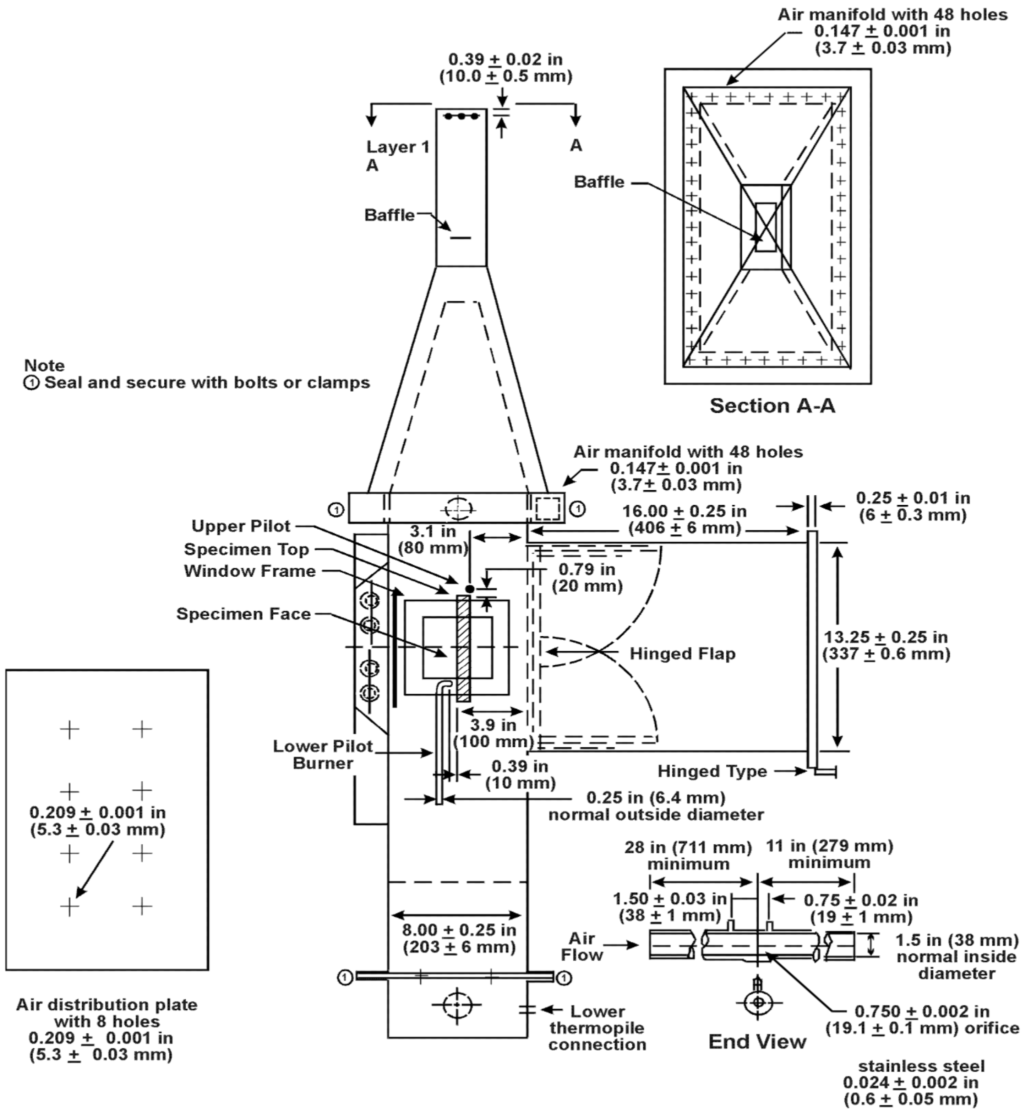
- **The absolute values of HR and HRR from the different calorimeters (reproducibility) were highly sensitive to airflow.**

The OSU Rate of Release Apparatus



Height =
102+6+787+310+30
+50+178 =1433 mm
=1.43 m

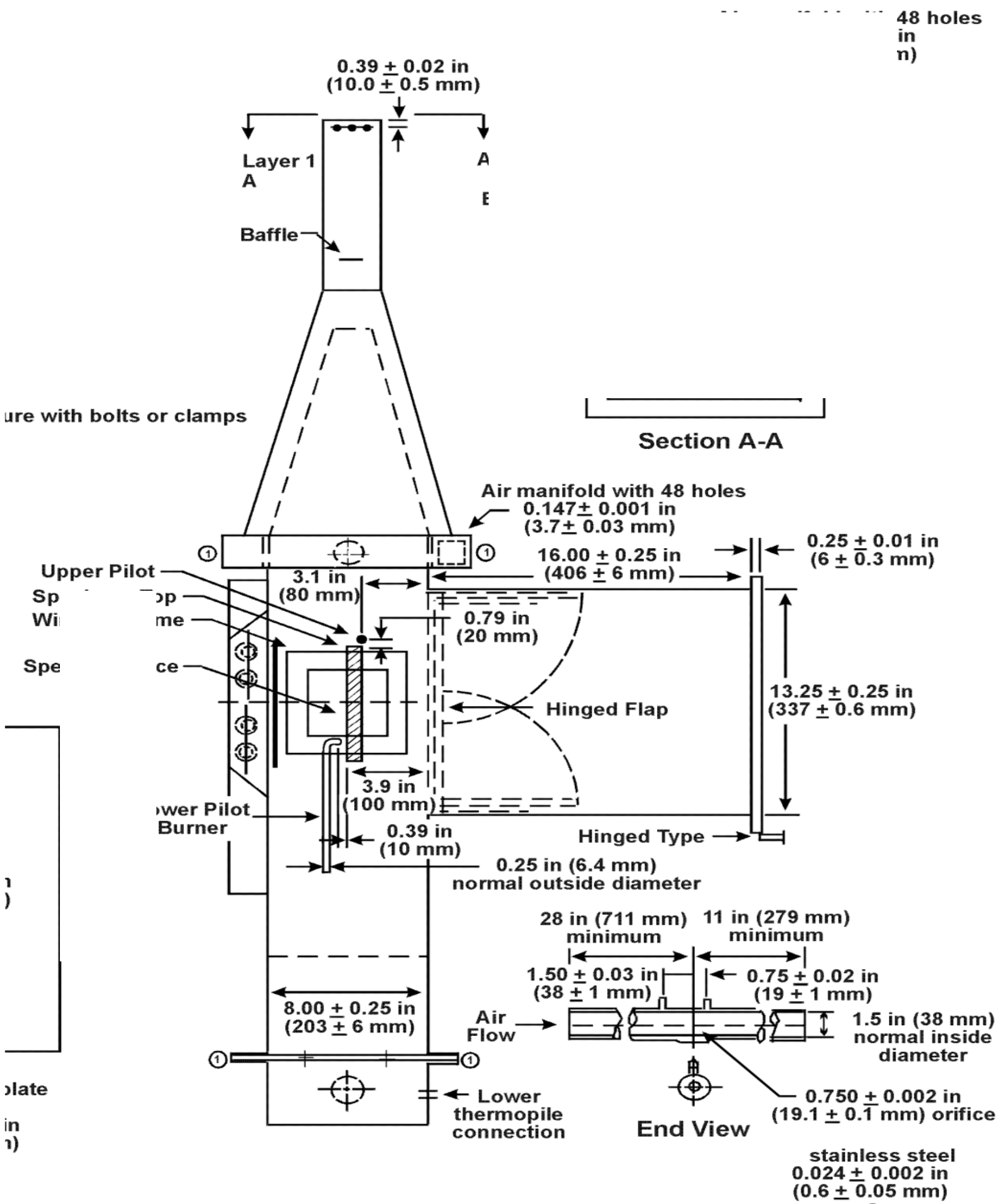
Front view



Side view

Physical characteristics of the FAA OSU Reactor

- Height: approximately 1.43 m
- Base dimensions: 0.406 m x 0.203 m
- Pressurized air supply from the base plate along with auxiliary air flow at the conical section
- Perforated plates near the entry region for air distribution
- Conical region before the chimney with secondary cooling flow
- Radiant heaters near the test specimen



Outline of the presentation

- Background and a bit of history
- Research areas and topics
- Progress to date, October 2022
- Accomplishments and Concluding remarks

Overall objective:

- **Develop a three-dimensional transient CFD model to simulate the turbulent flow conditions in the OSU Rate of Release reactor**

  …….

and to predict the rate of heat release from a sample in the presence of convective and radiative heat transfer conditions.

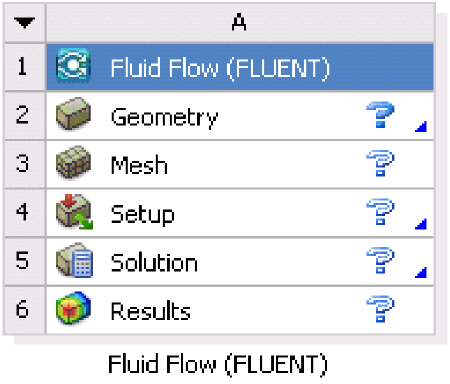
*Computational tool: **ANSYS FLUENT***

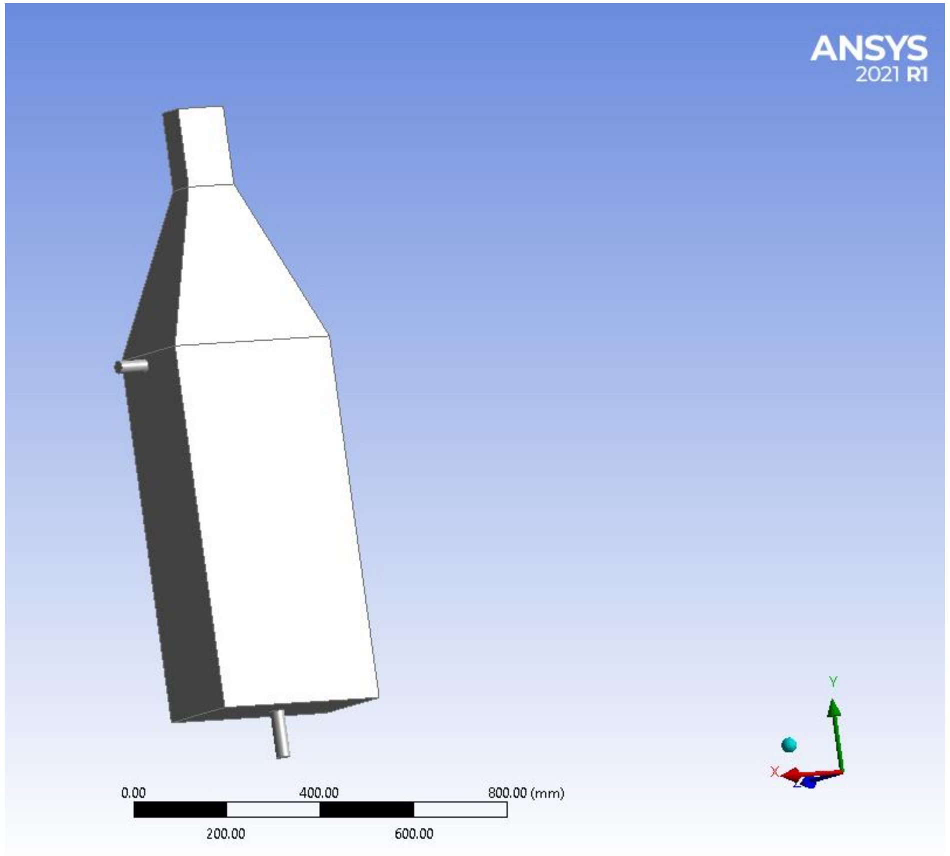
- ANSYS-FLUENT is an advanced general-purpose multiphysics simulation software package
- The FLUENT code originated from in the 1980s (UK) that became the dominant market leader by the late 90s
- ....and is today part of ANSYS Inc., one of the leading simulation software providers for engineering.
- The package continues to expand on possibilities for the calculation of many complex, real world problems
- ANSYS-FLUENT is used by the automobile, aerospace, construction, ***military*** , manufacturing, and bioengineering industries.

Outline of the presentation

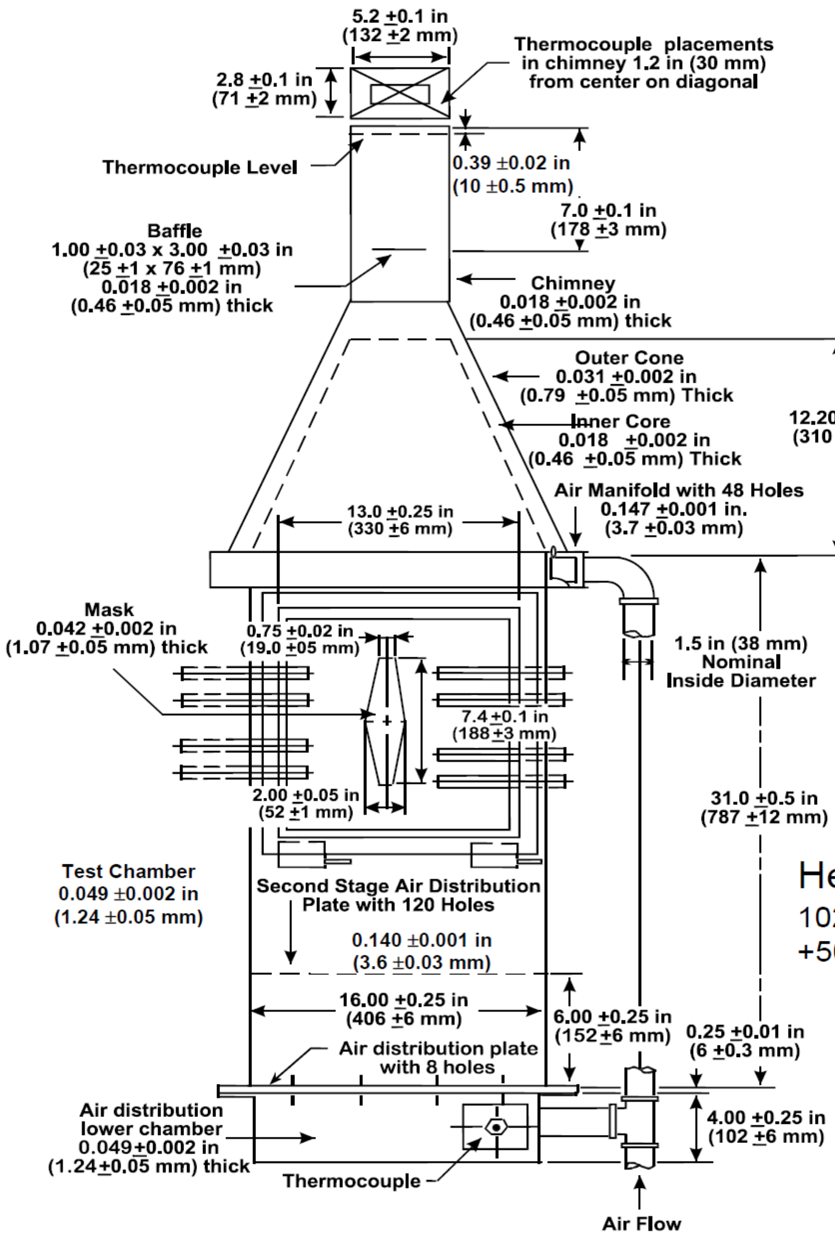
- Background and a bit of history
- Research areas and topics
- Progress to date *(since summer 2021)*
- Concluding remarks

- ✓ ANSYS DesignModeler application
- ✓ The outer shell of the OSU reactor created using Geometry
- ✓ Geometry was meshed using FLUENT Mesh
- ✓ Transient 3D turbulent flow problem was formulated using Setup
- ✓ Simulations were performed – using Solution
- ✓ Postprocessing of Results

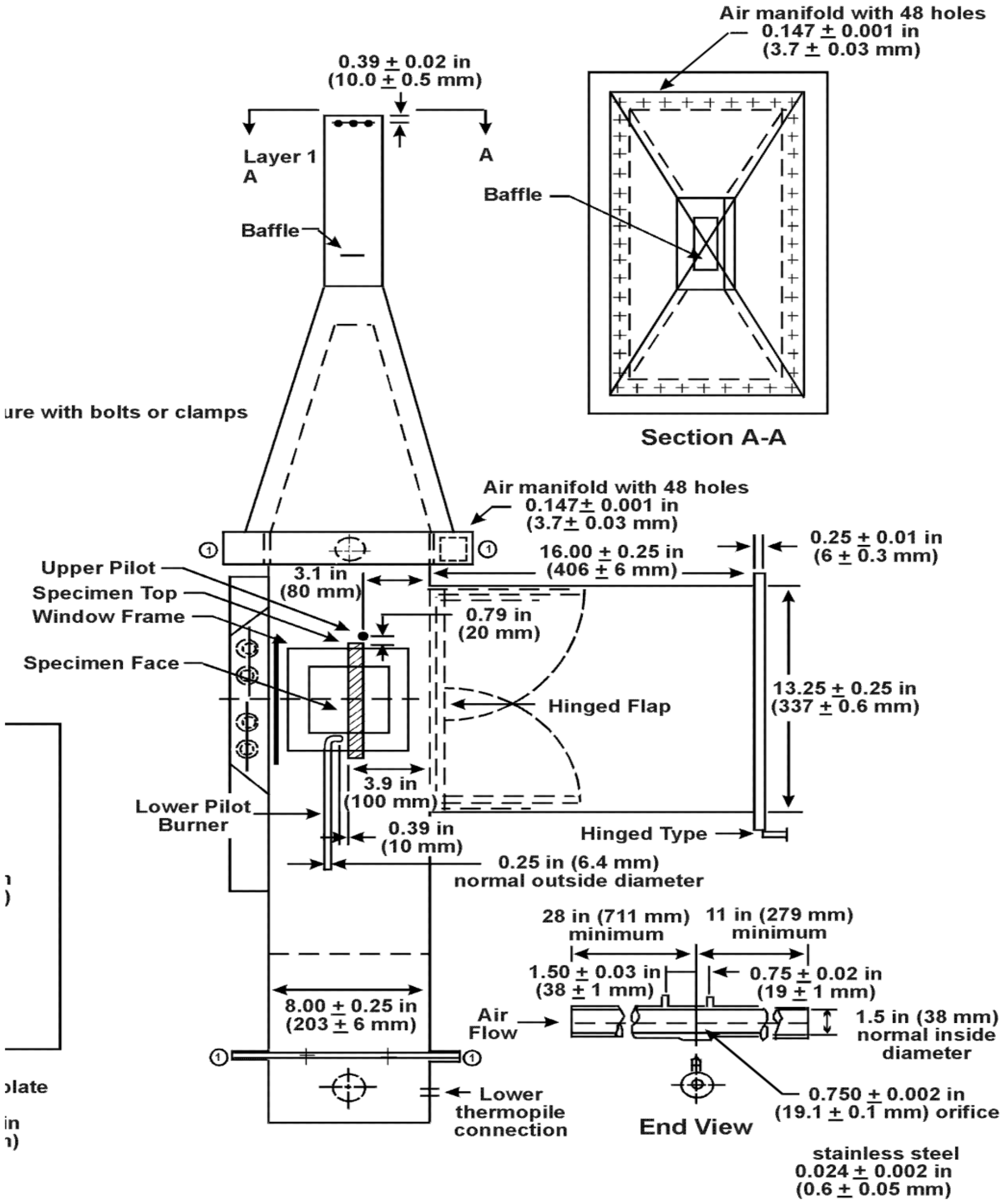


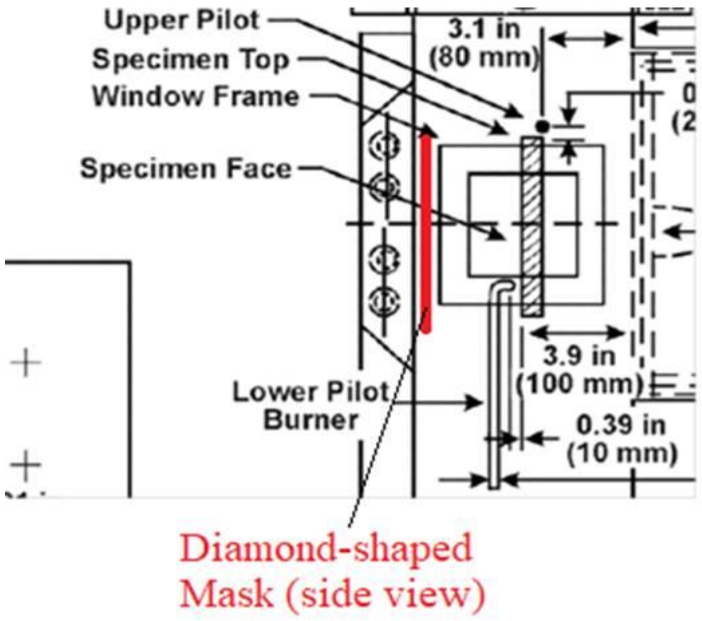


The inner details of the OSU reactor



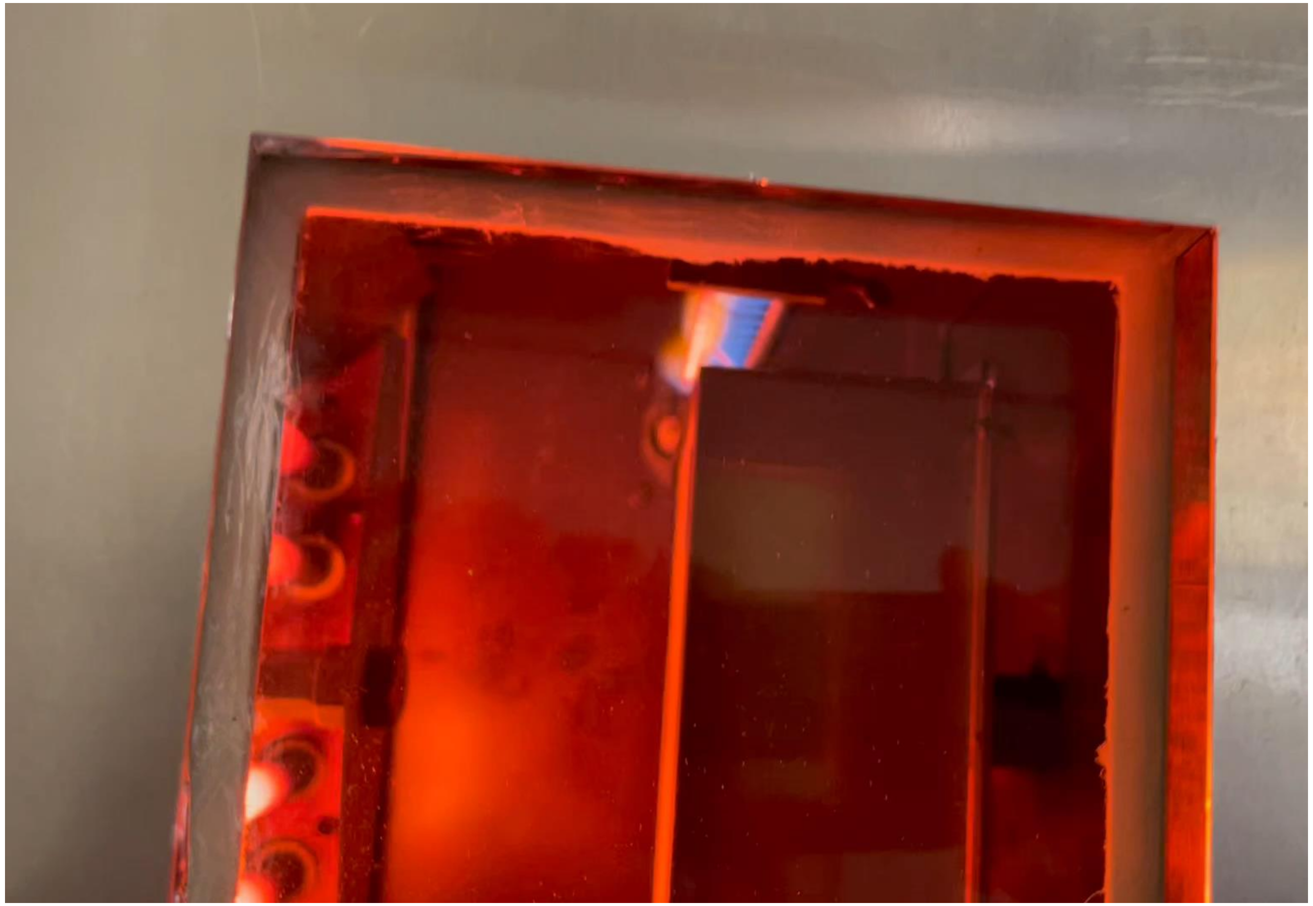
Height =
102+6+787+310+30
+50+178 =1433 mm
=1.43 m





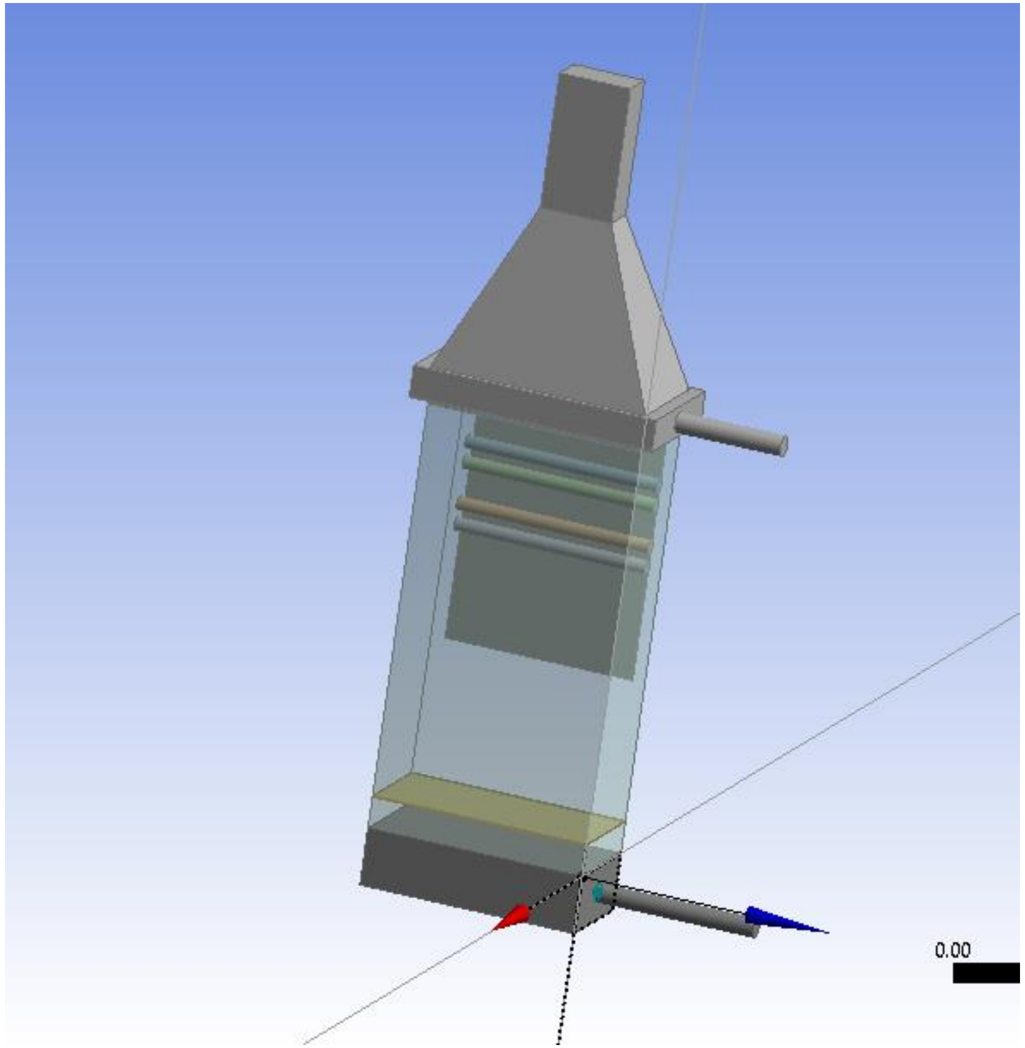
Upper and lower pilot burners:

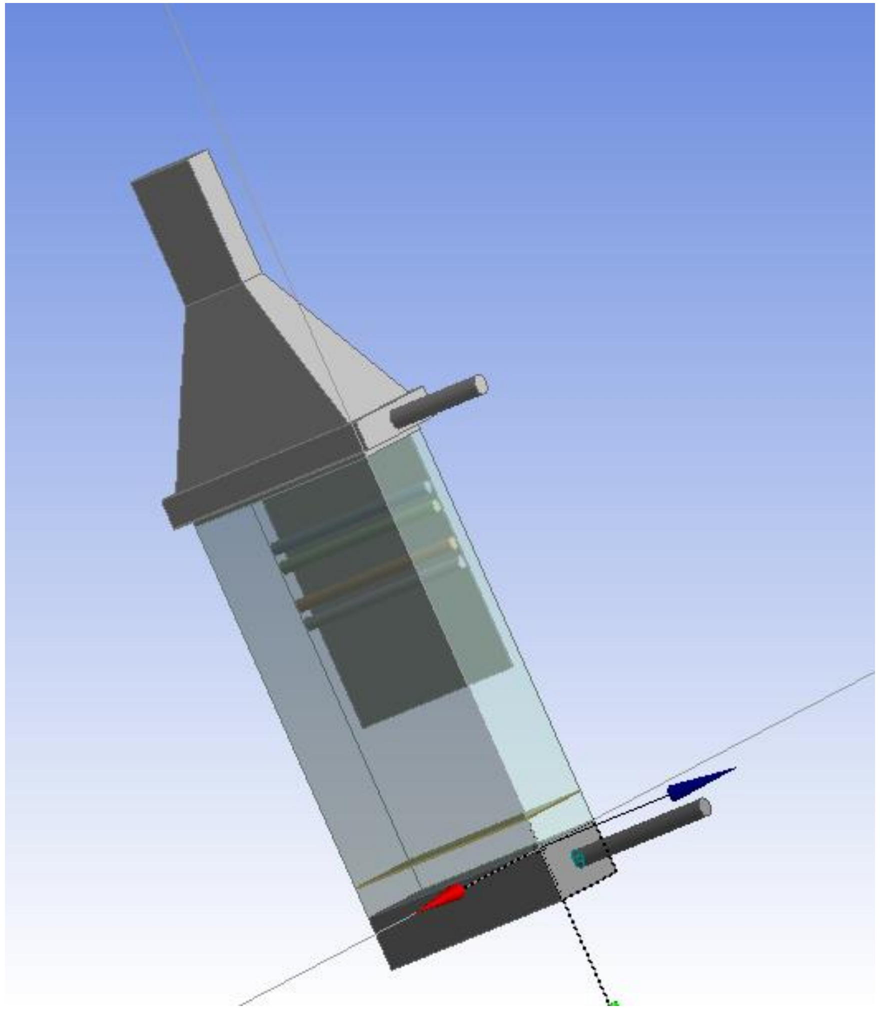
- to burn the combustible gases coming out of the sample (pyrolysis)
- Consider pyrolysis and combustion in ‘future models’

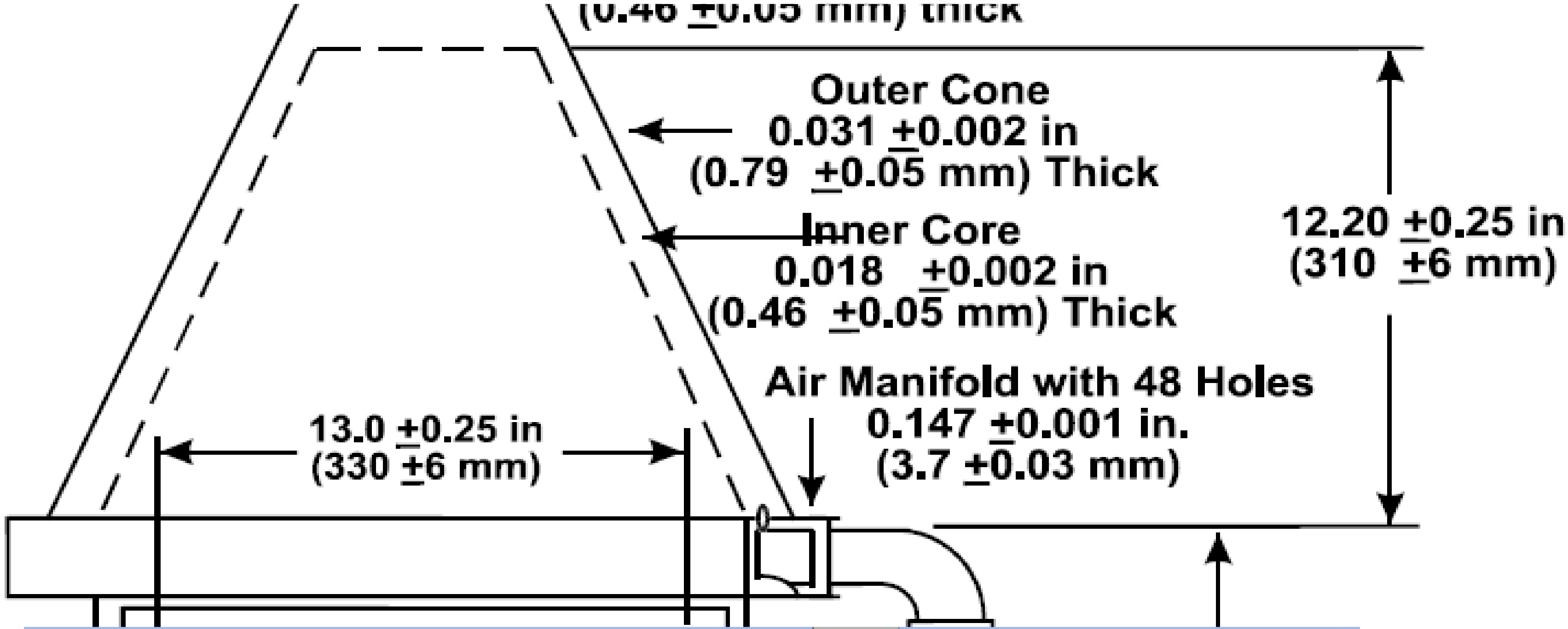


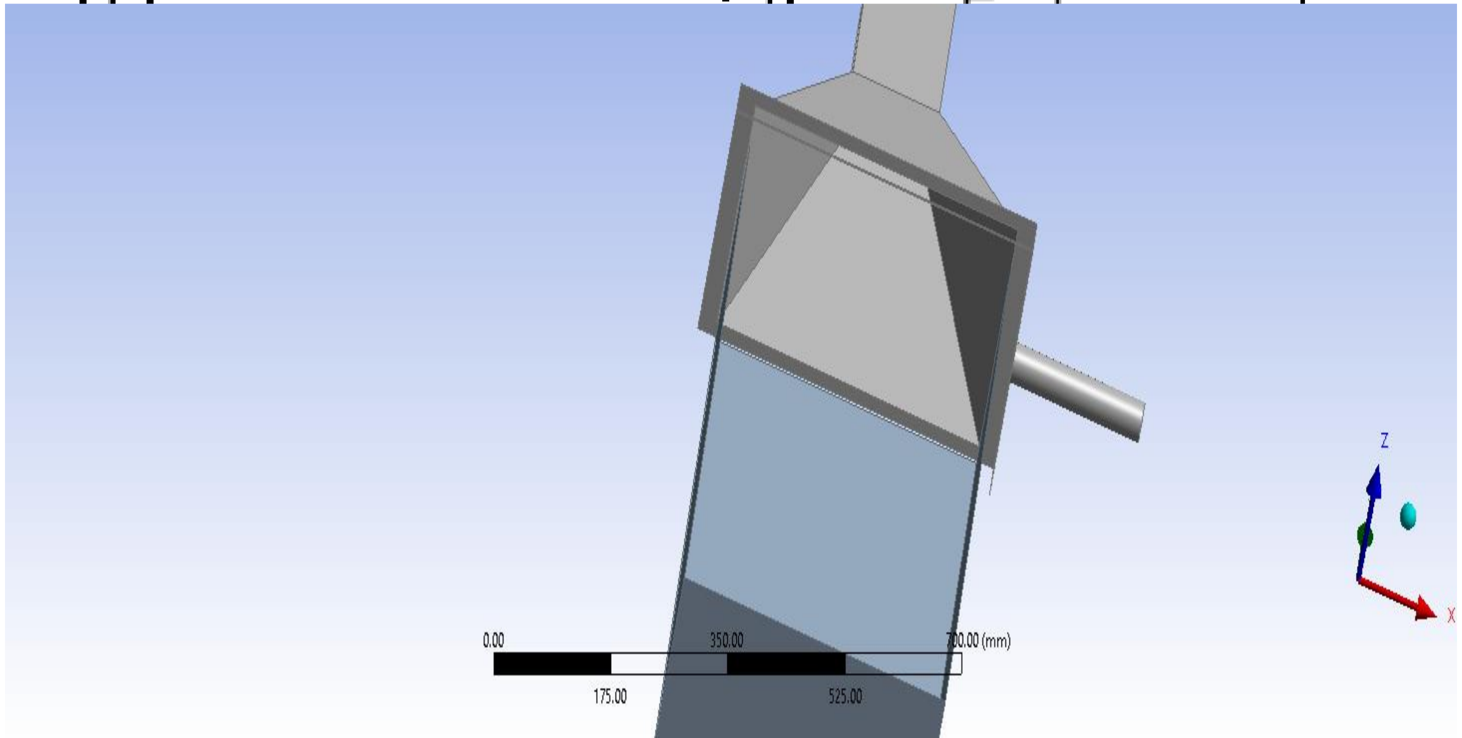
Radiant heat transfer from the cylindrical heaters

Preliminary DesignModeler application for the OSU reactor

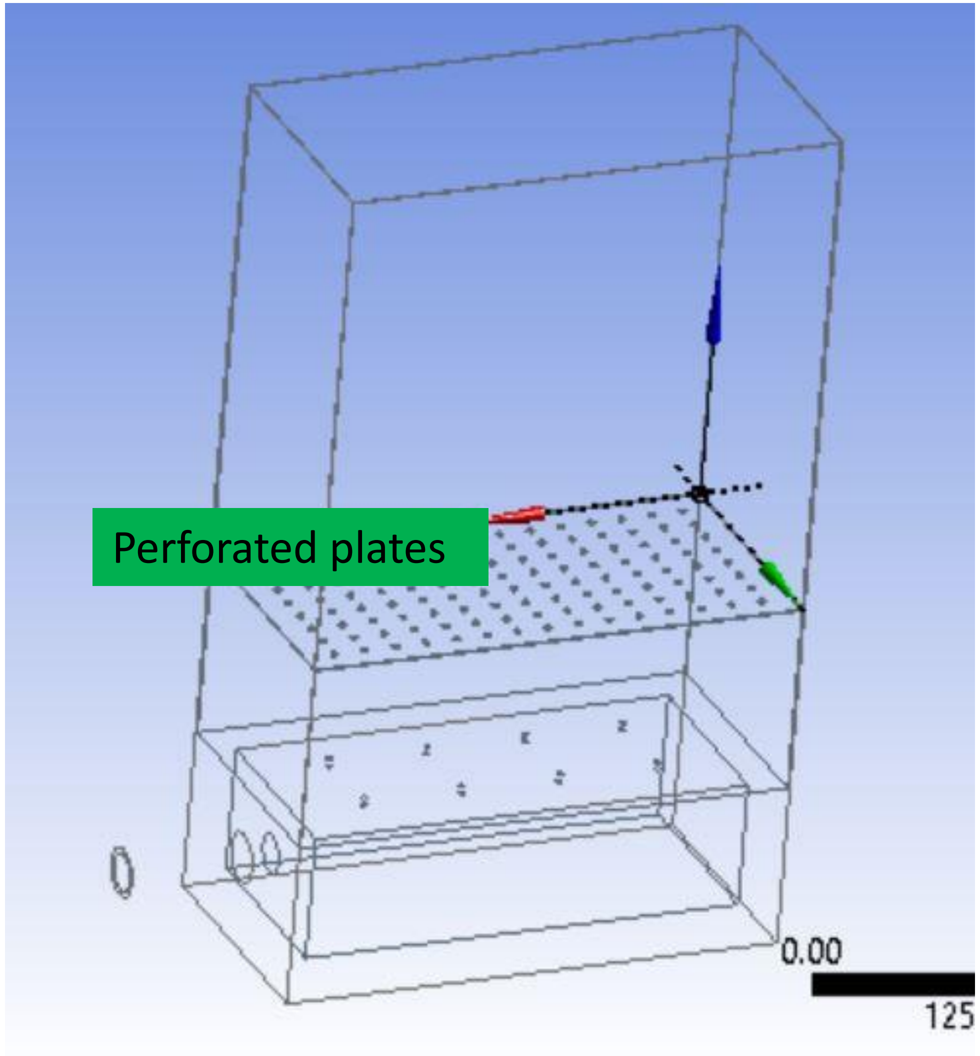


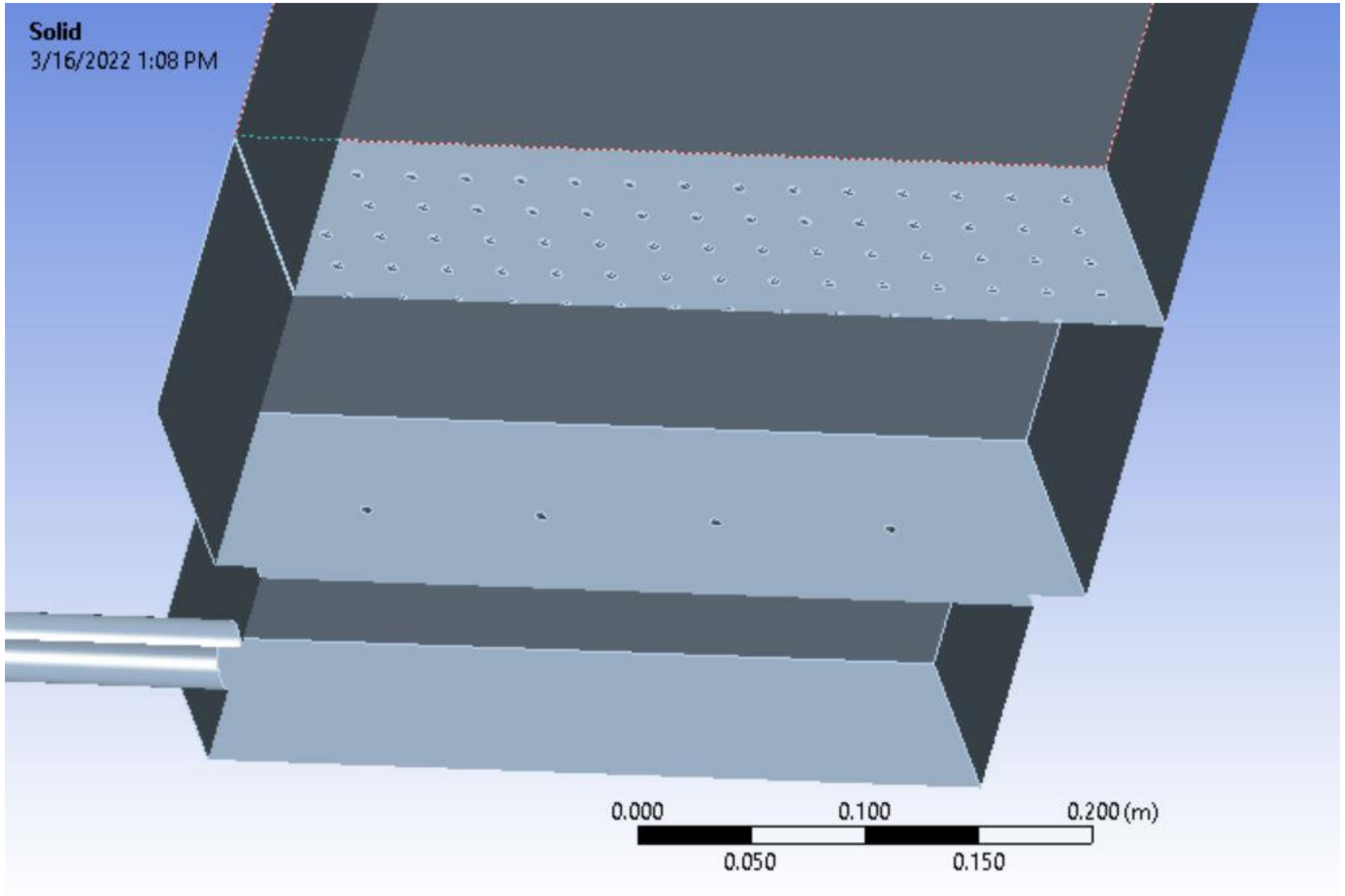



(0.46 ±0.05 mm) thick
Outer Cone
0.031 ±0.002 in
(0.79 ±0.05 mm) Thick
Inner Core
0.018 ±0.002 in
(0.46 ±0.05 mm) Thick
12.20 ±0.25 in
(310 ±6 mm)
Air Manifold with 48 Holes
0.147 ±0.001 in.
(3.7 ±0.03 mm)
13.0 ±0.25 in
(330 ±6 mm)

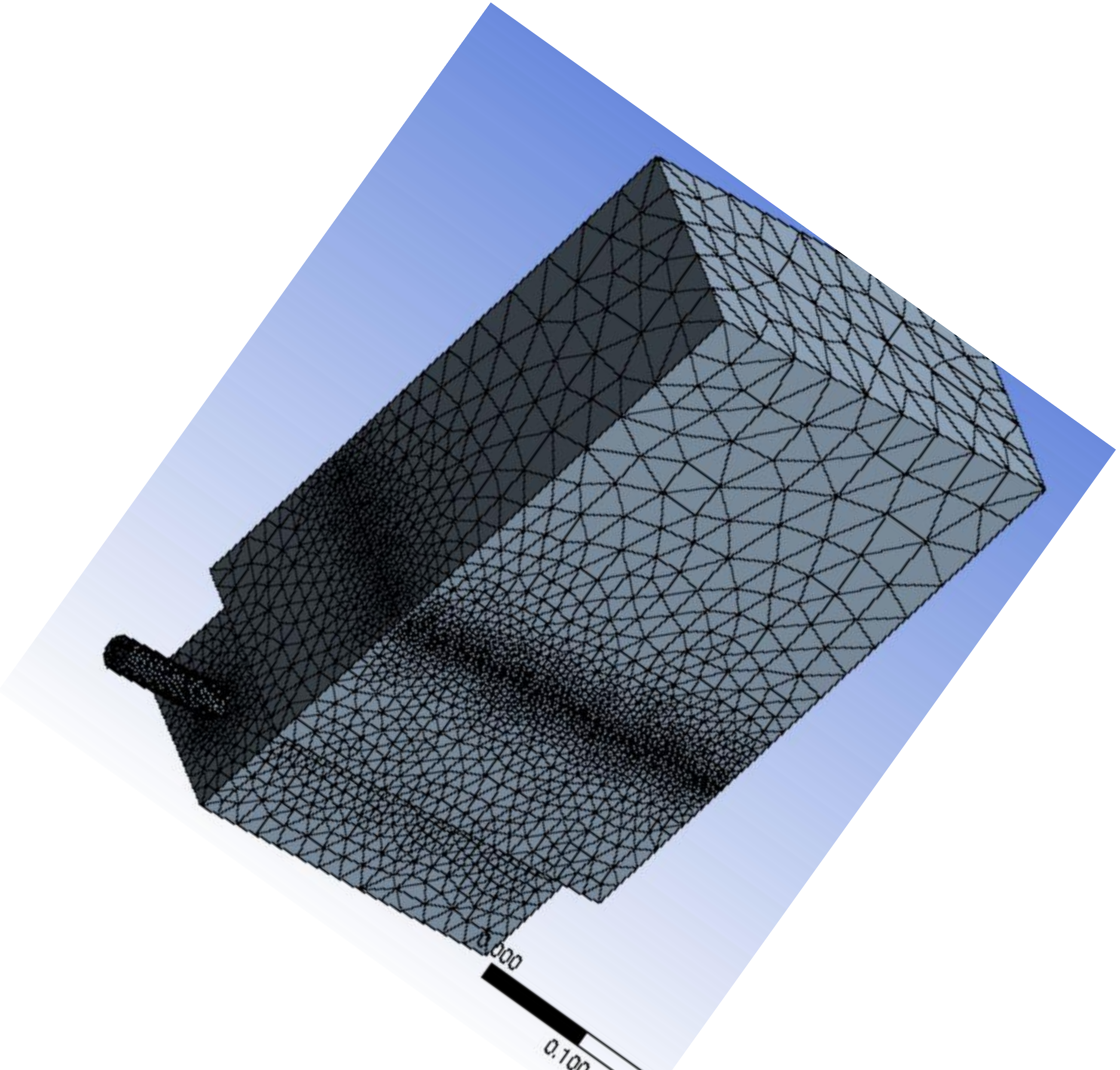

0.00
350.00
700.00 (mm)
175.00
525.00
Z
X

Entry section geometry



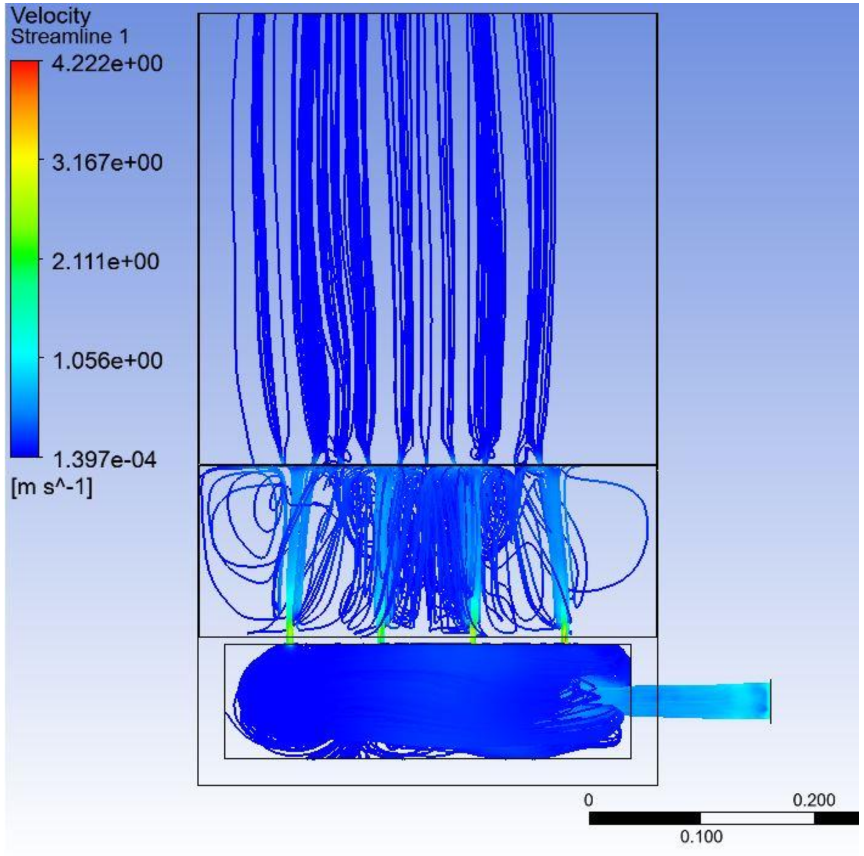


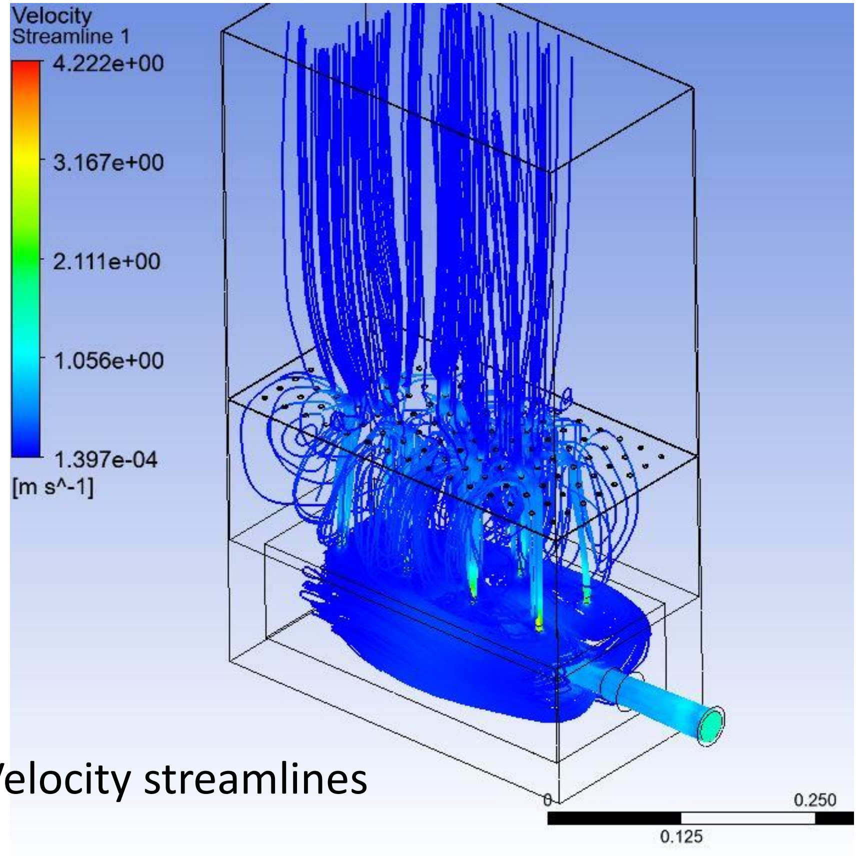
Mesh generation including the perforated plates



Entry section fluid dynamics results

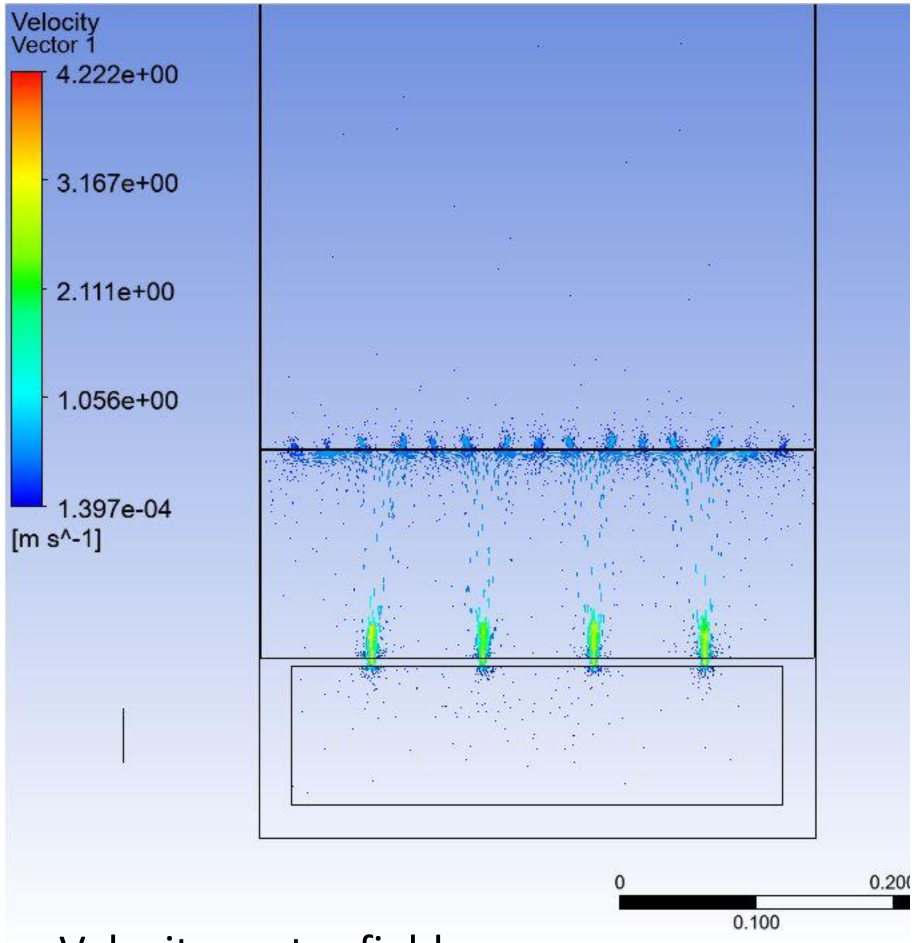
- Inlet velocity - 1.5 m/s
- Outlet pressure - 1 atm



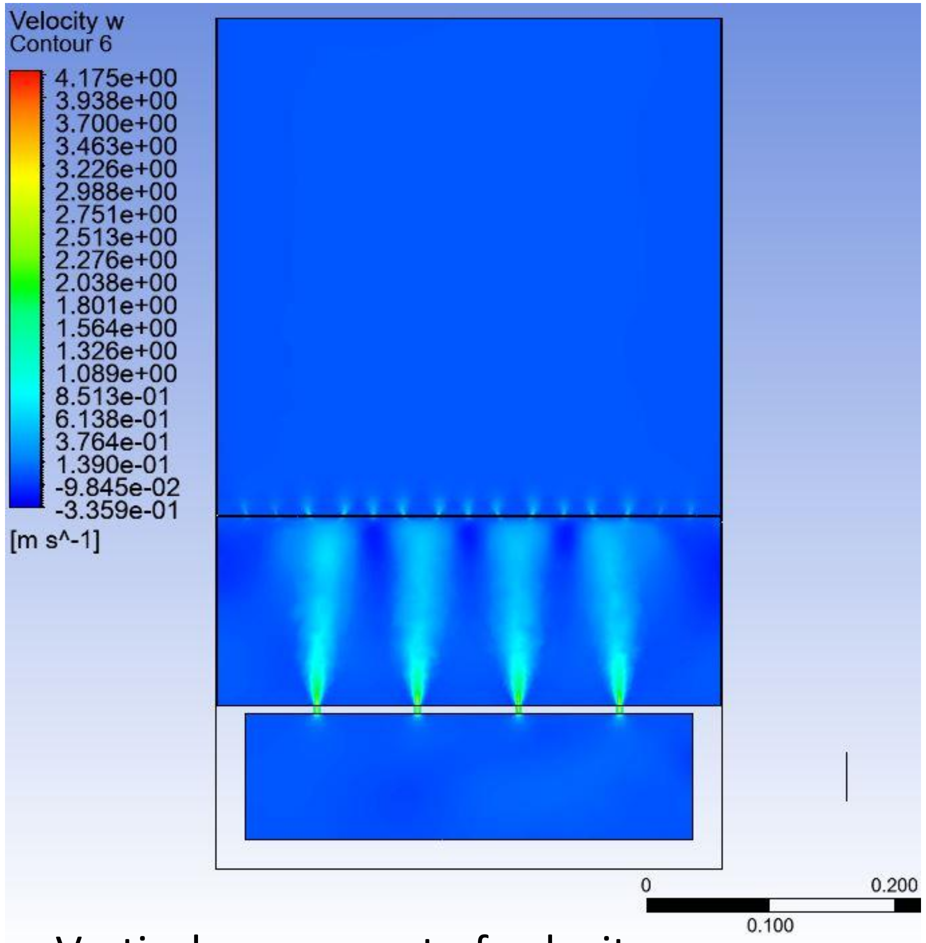


Velocity streamlines

Entry section fluid dynamics

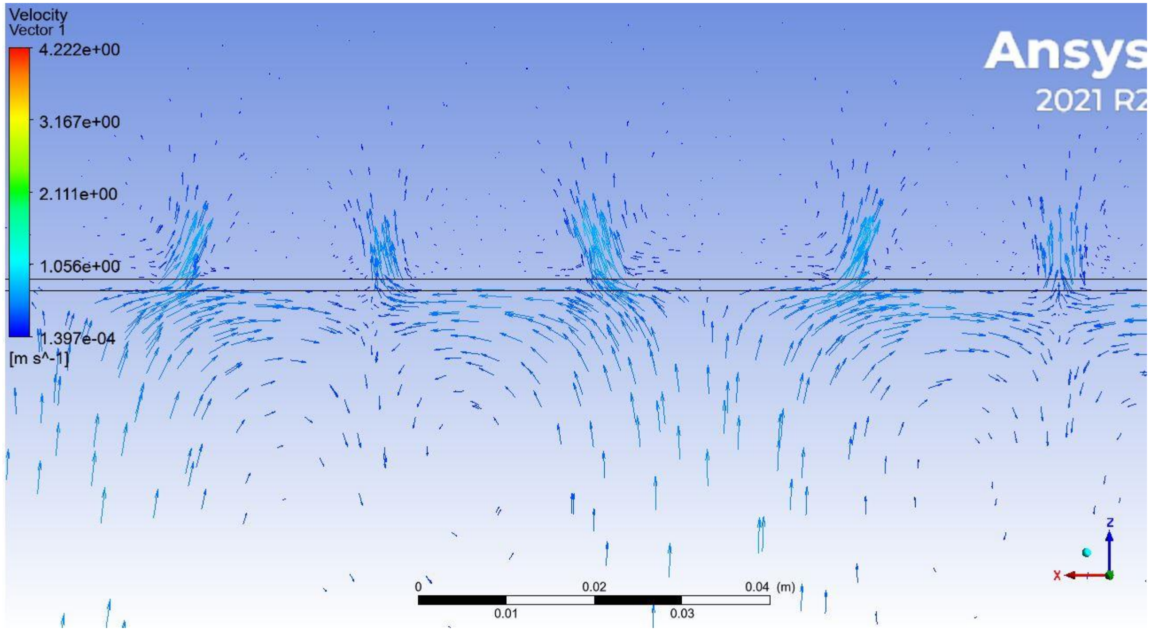


Velocity vector field

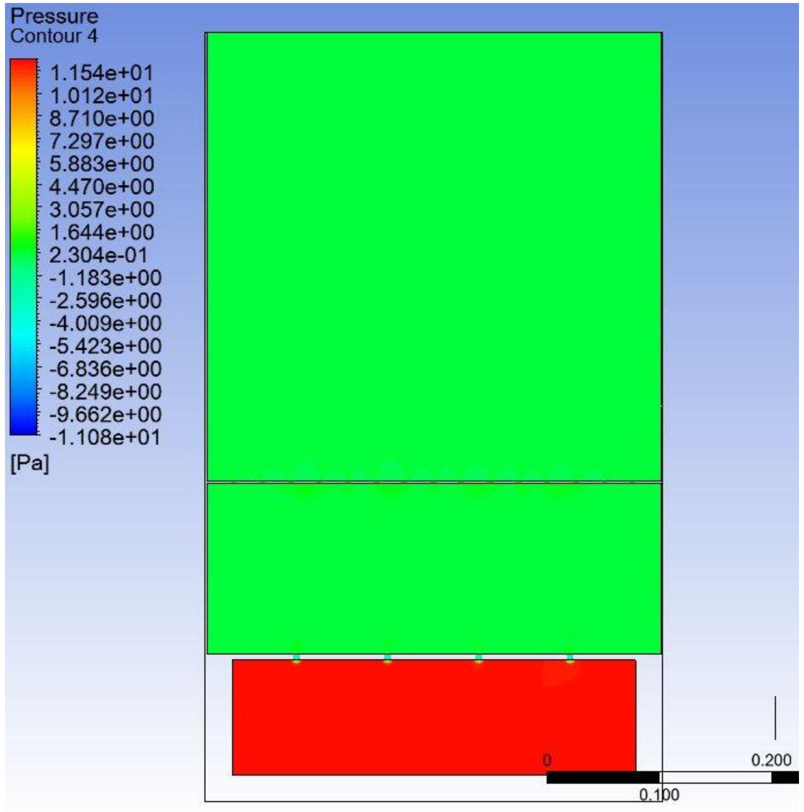


Vertical component of velocity

Entry section fluid dynamics results…

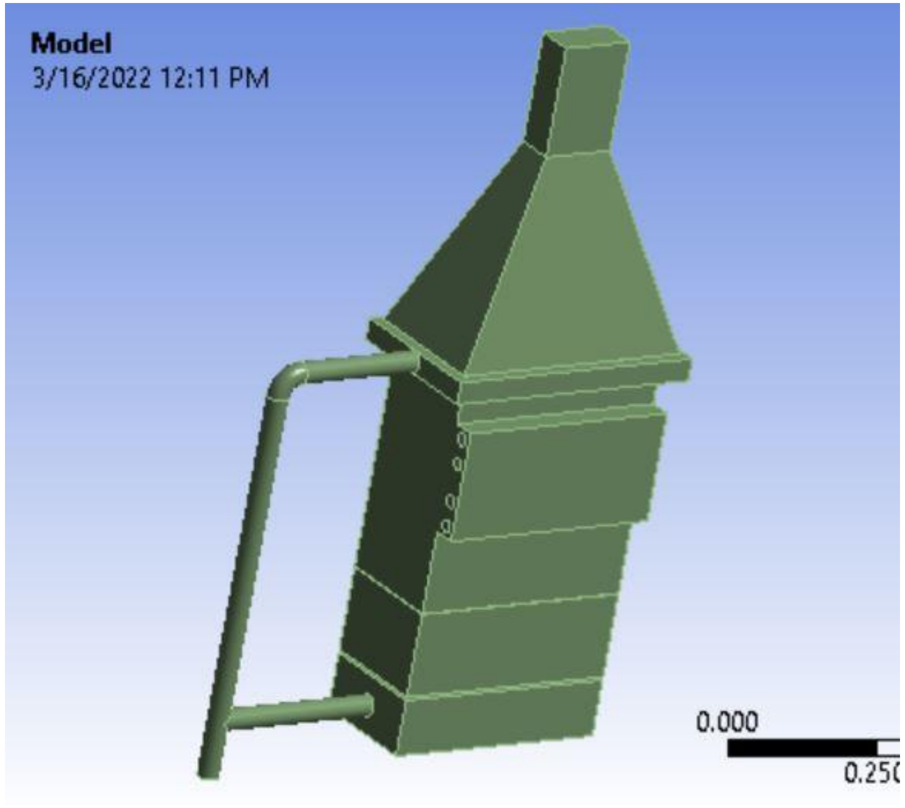


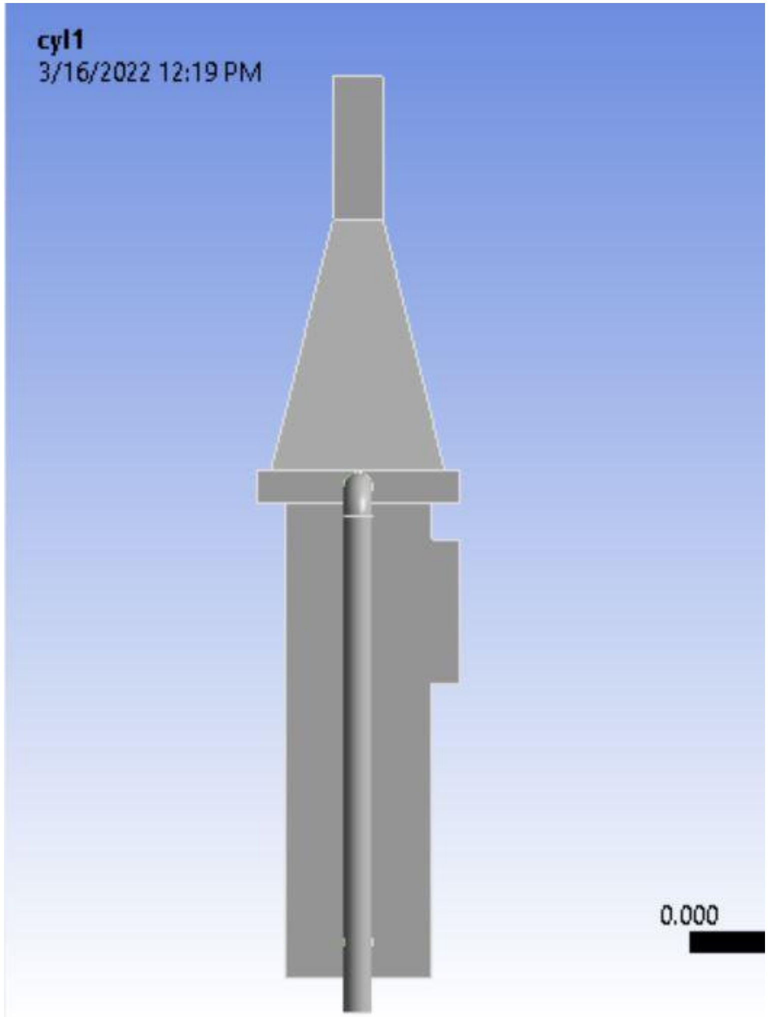
Velocity vector field at upper perforated plate



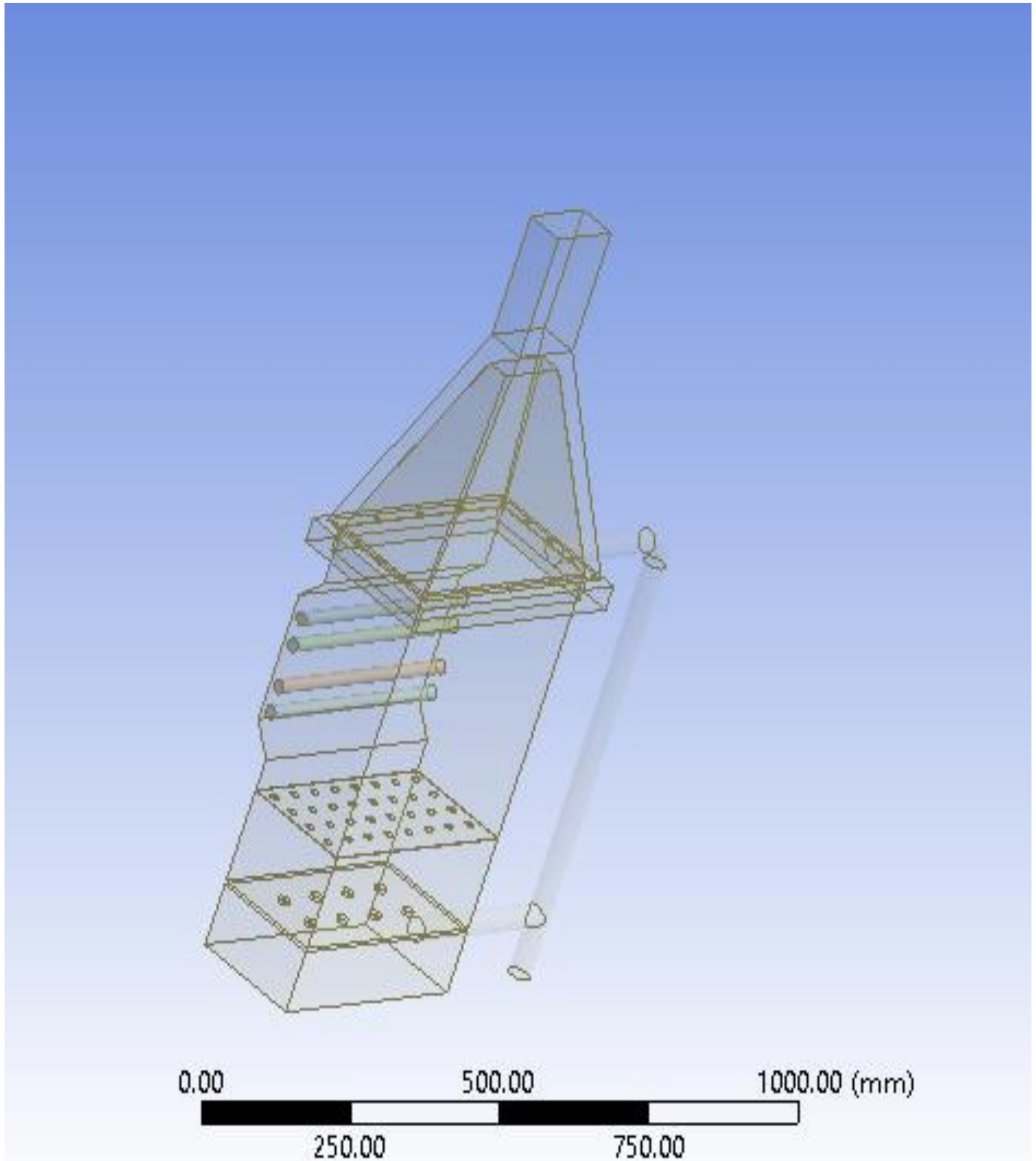
Pressure field

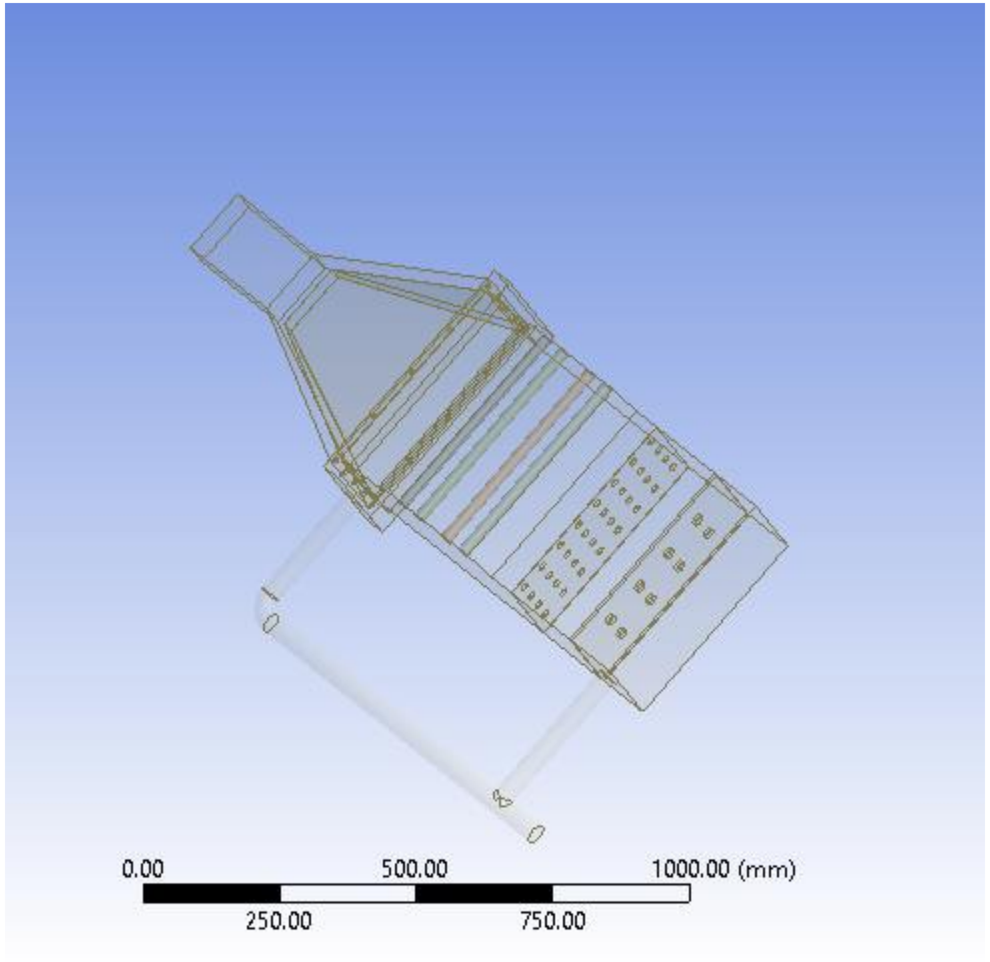
Modeled OSU calorimeter geometry



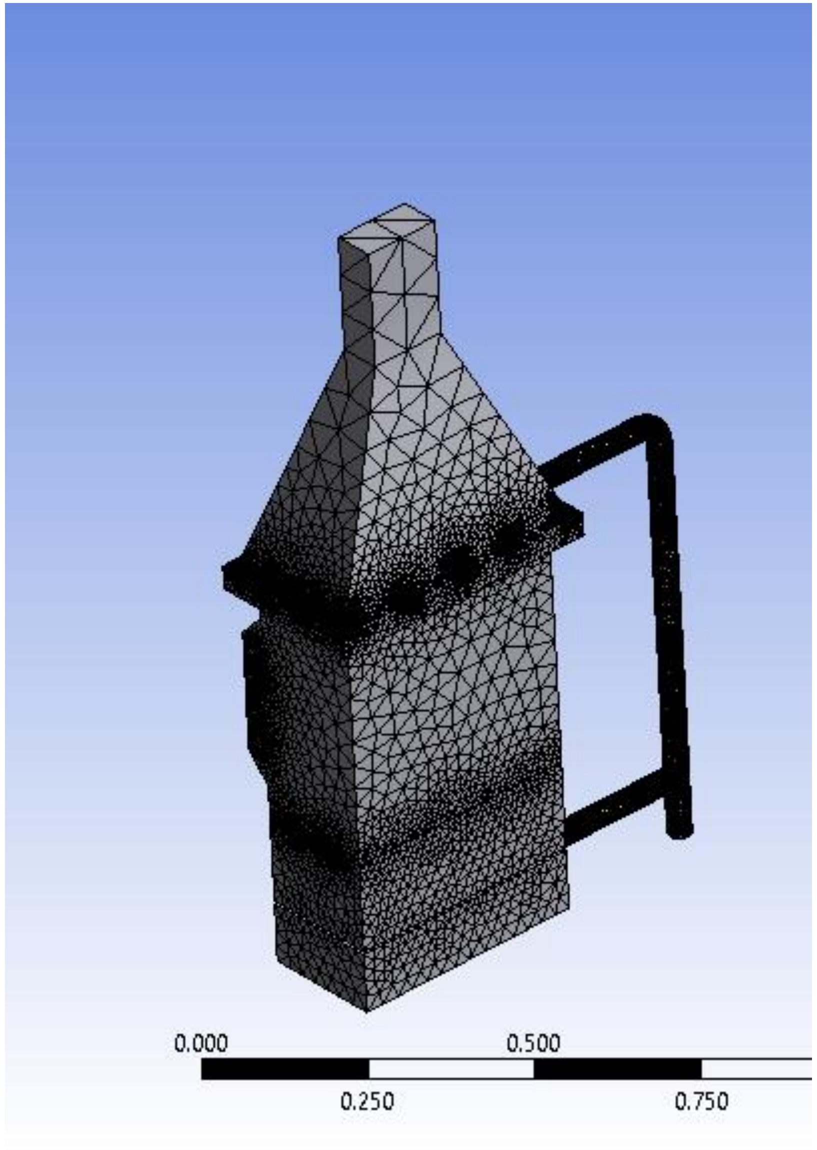


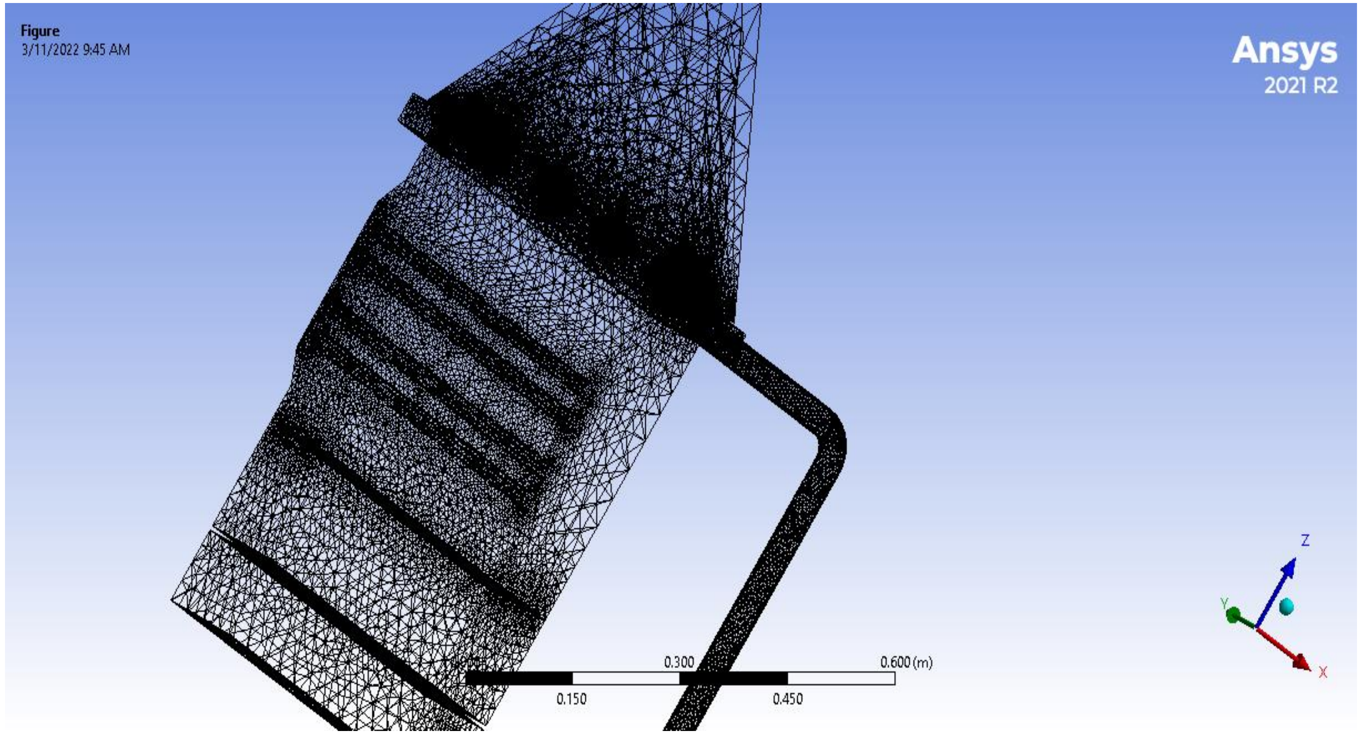
Internal details of the modeled OSU reactor



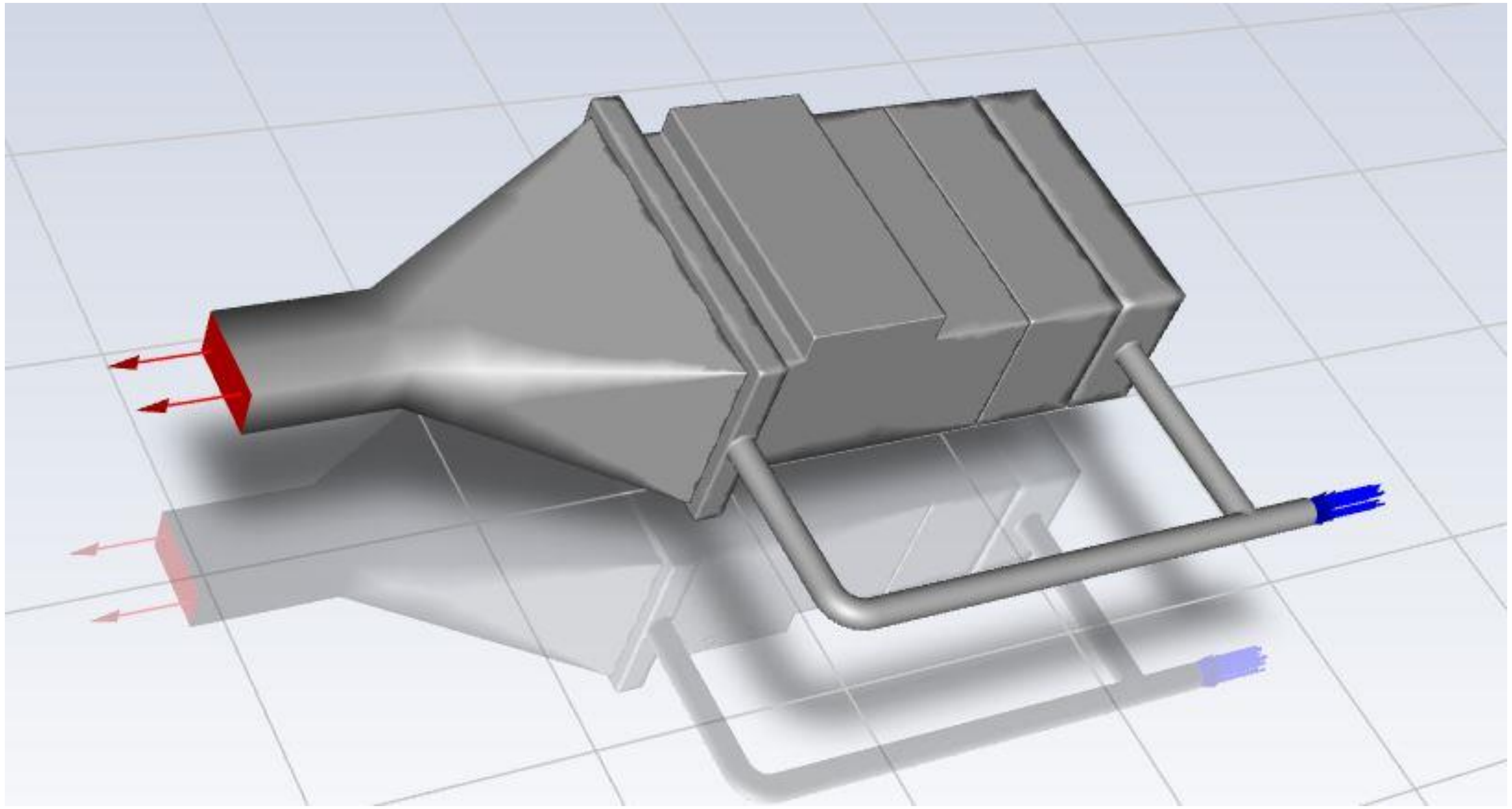


Mesh generation with internal hardware





OSU Calorimeter
(for fluid flow and heat transfer studies)

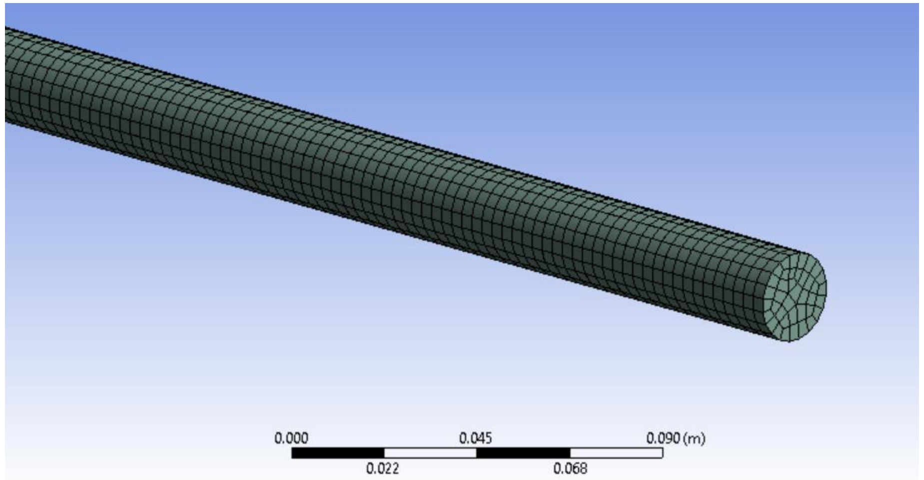


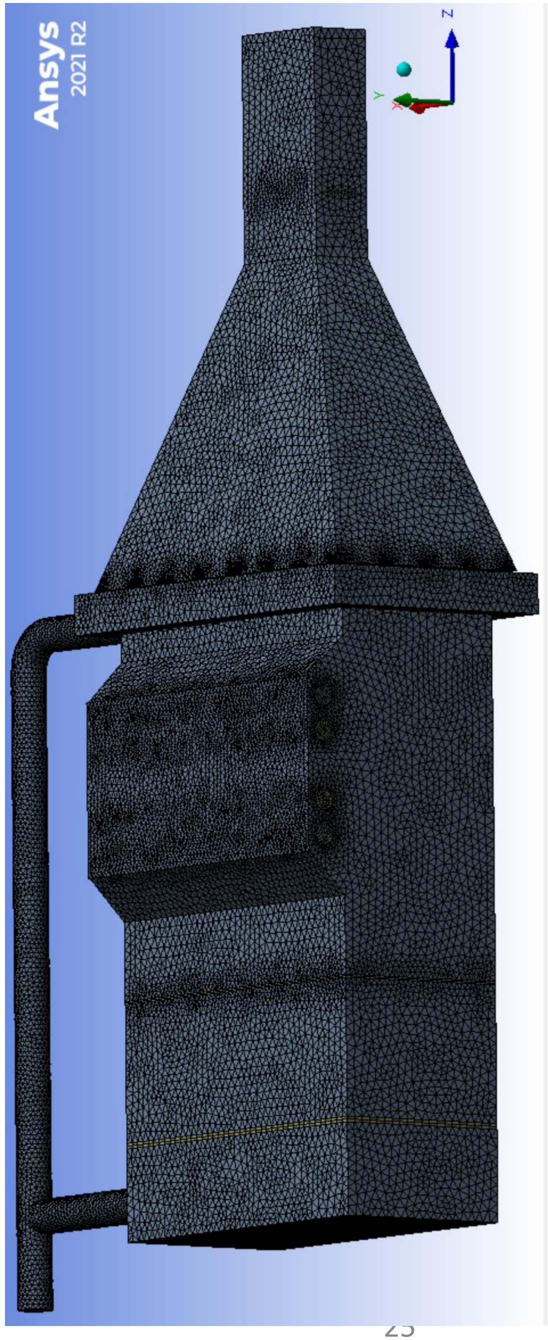
Meshing

- Combination of tetrahedral and hexahedral cells
- Conformal mesh
- Automatically generated in Fluent
  - Number of cells:
  - Number of nodes:

<u>Radiant Cylinders</u>

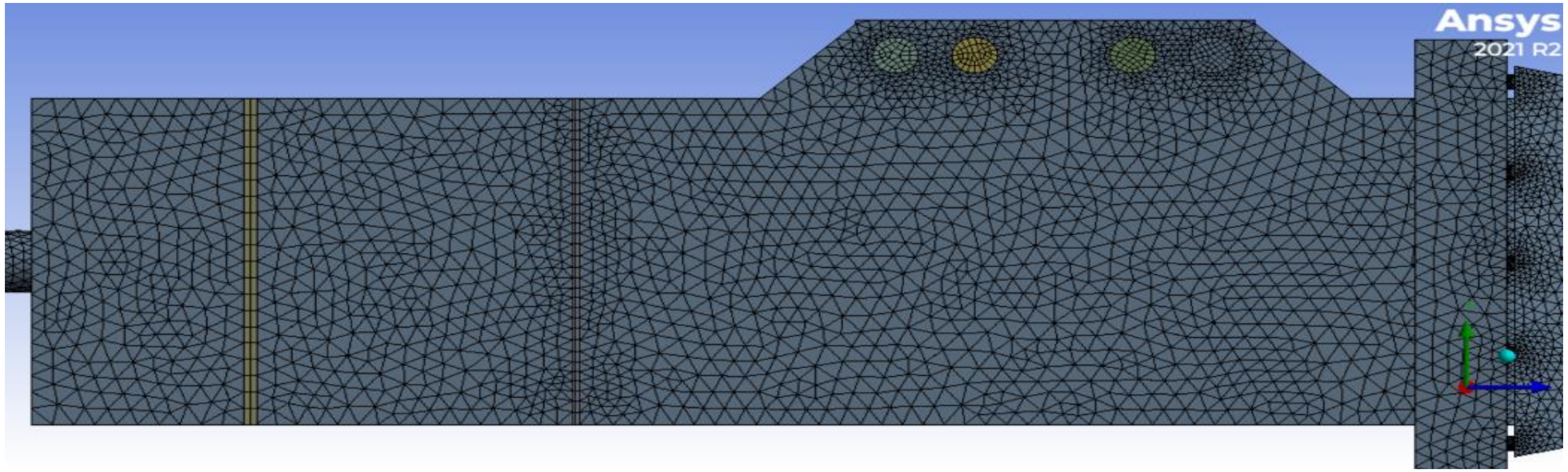
Hexahedral cells
Generated automatically in Fluent meshing

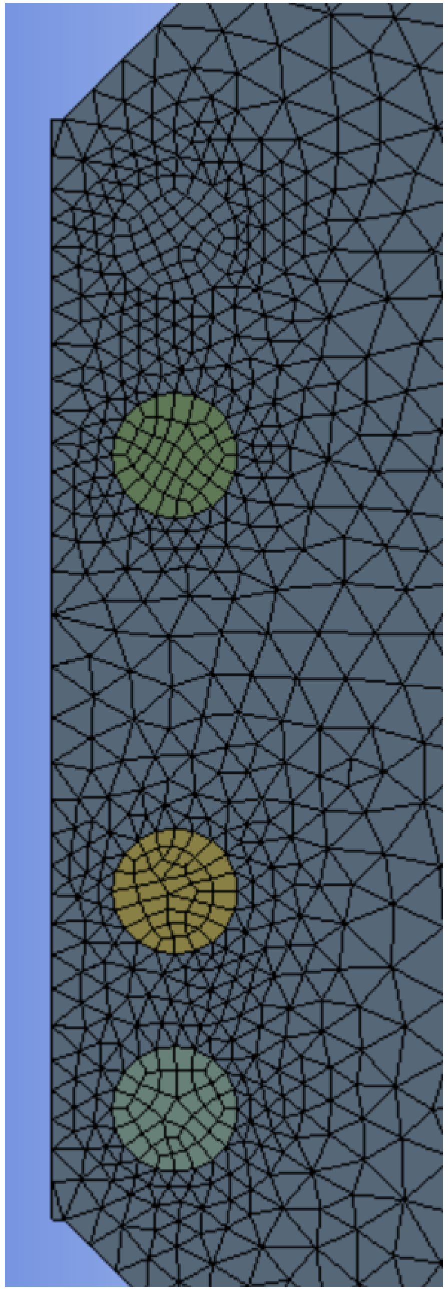




Conformal meshing

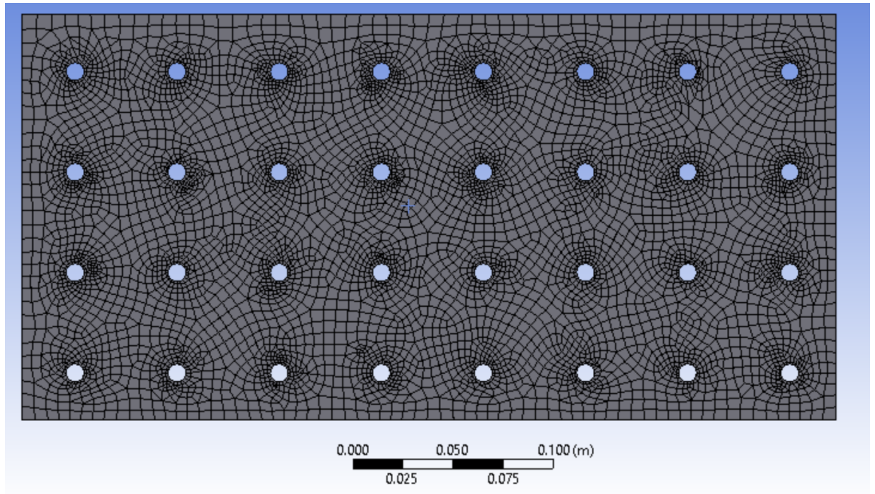
- Conformal meshing of cylinders and plates in fluid
- Locations of nodes of either side of interfaces match

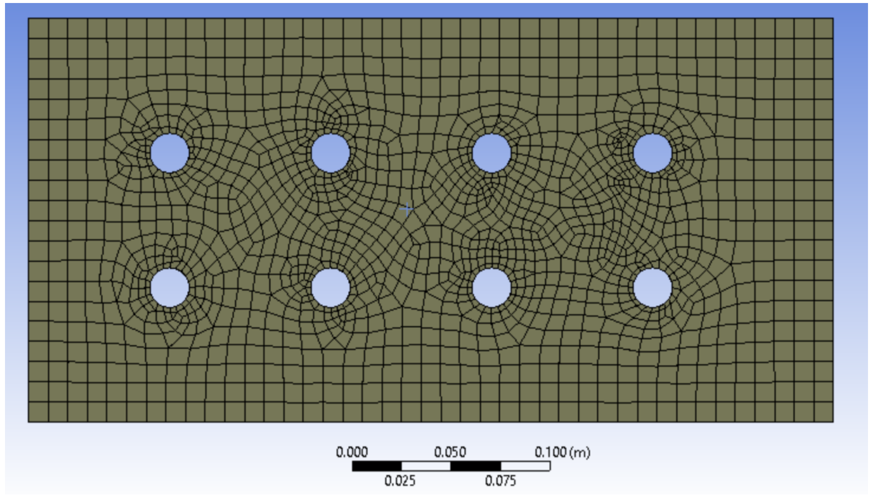




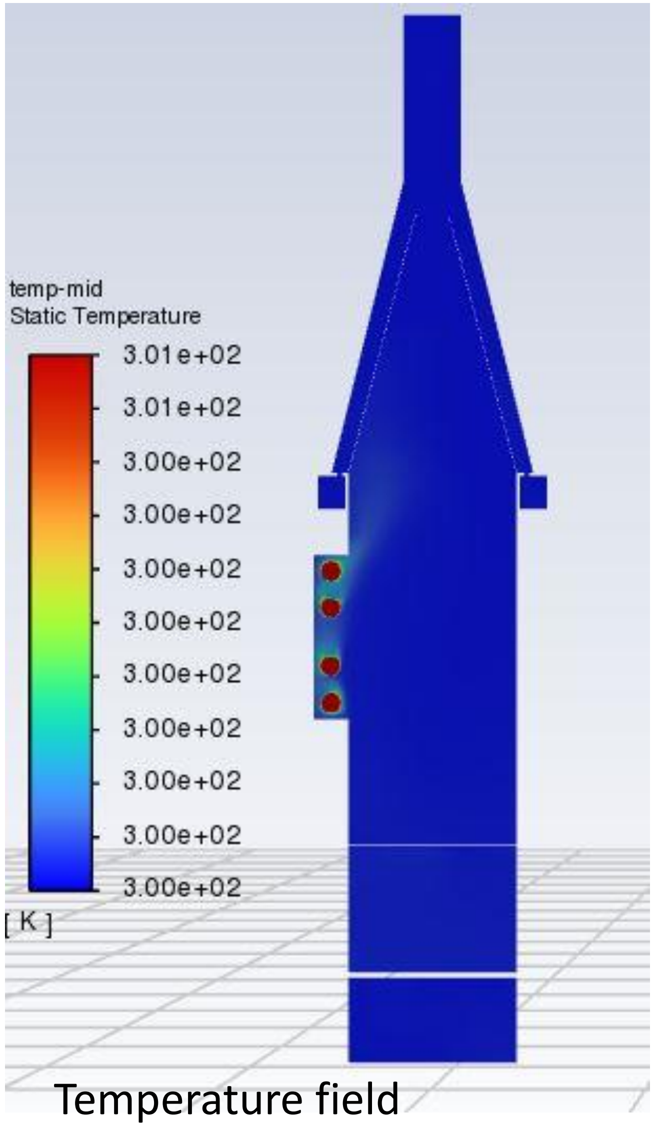
Meshing of perforated plates

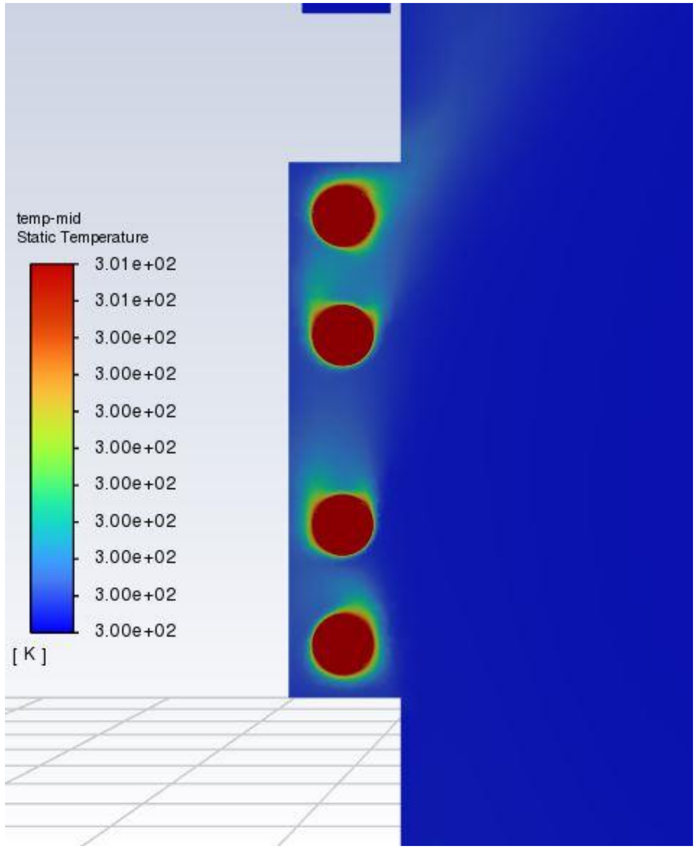
- Unstructured mesh, automatically generated
- Hexahedral cells
- Simplification of geometry to reduce computation time
  - Reduced number of holes in plates
  - Increased hole diameters





OSU device – preliminary results



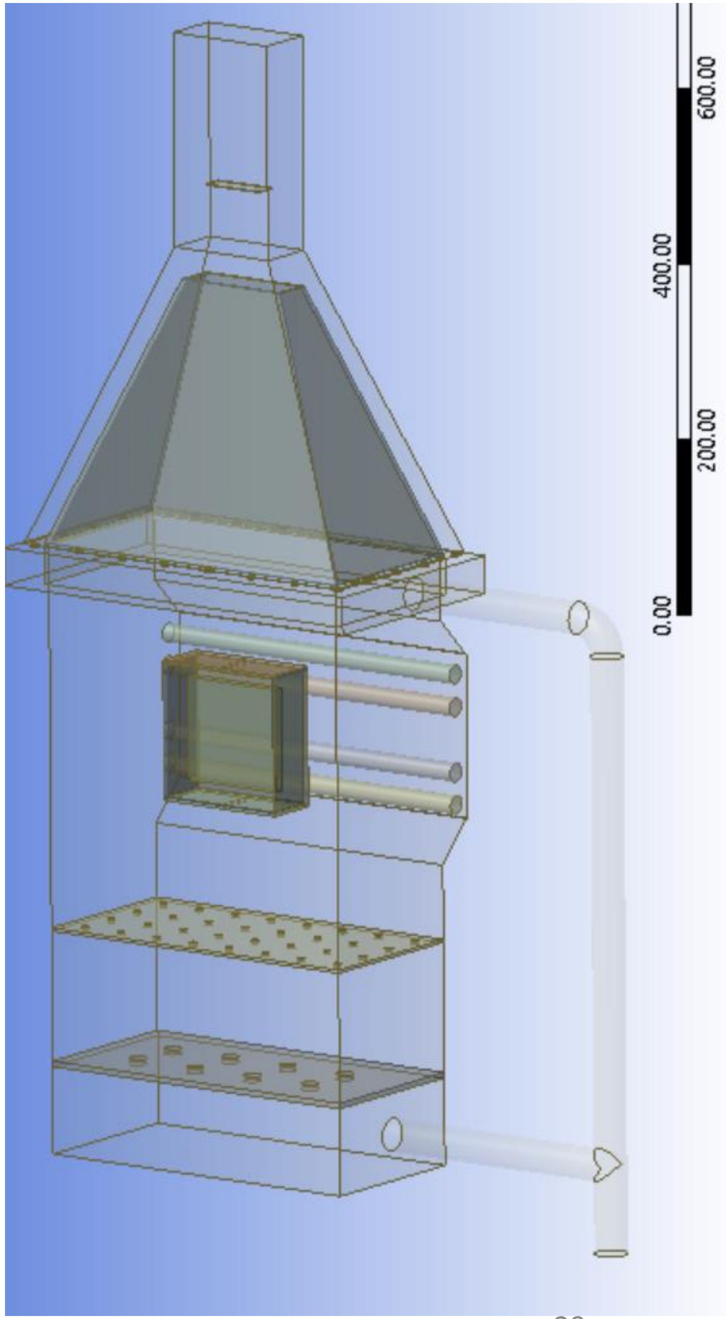


Temperature field
(early timestep in transient solution)

- **Buoyant Convection from heated cylinders**
- **Total 1000 W volumetric heat source**

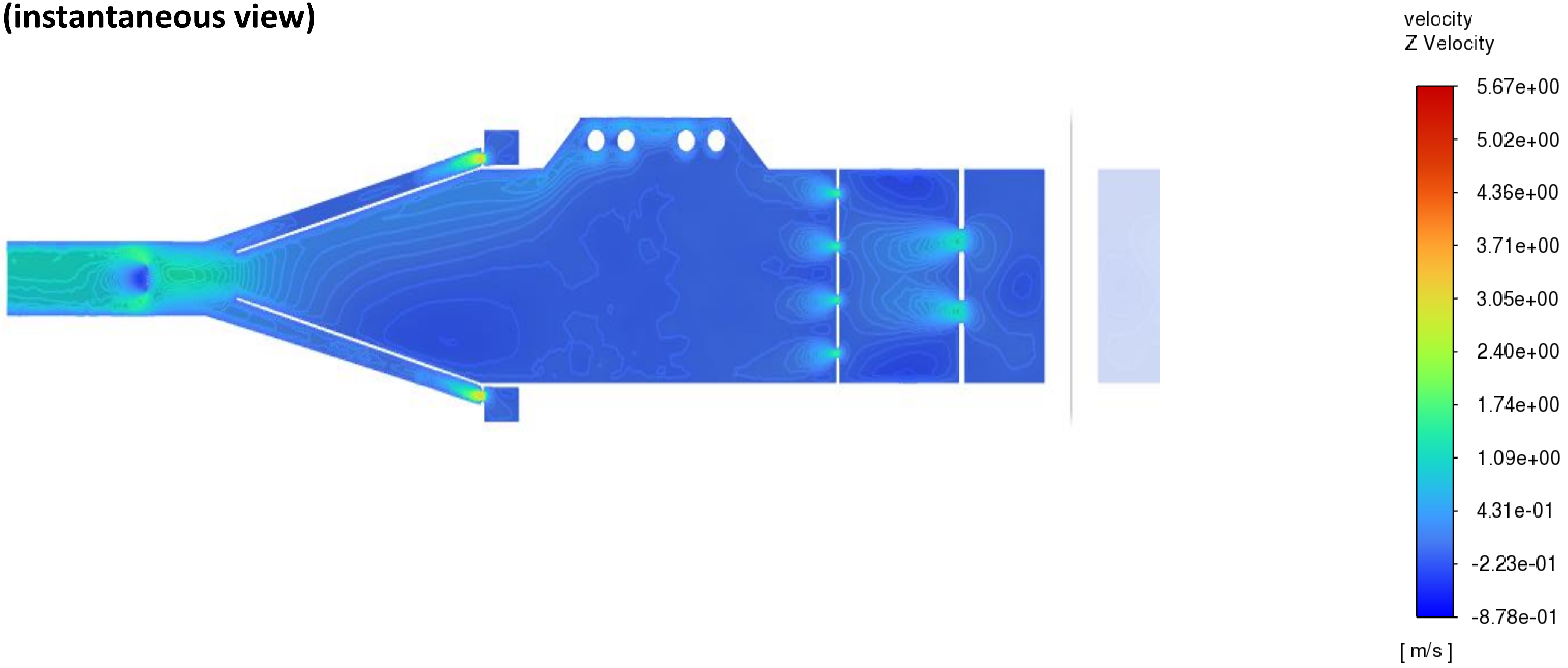
Addition of specimen holder and specimen

- Specimen modeled as volumetric heat source
- Holes in top and bottom of specimen holder for airflow

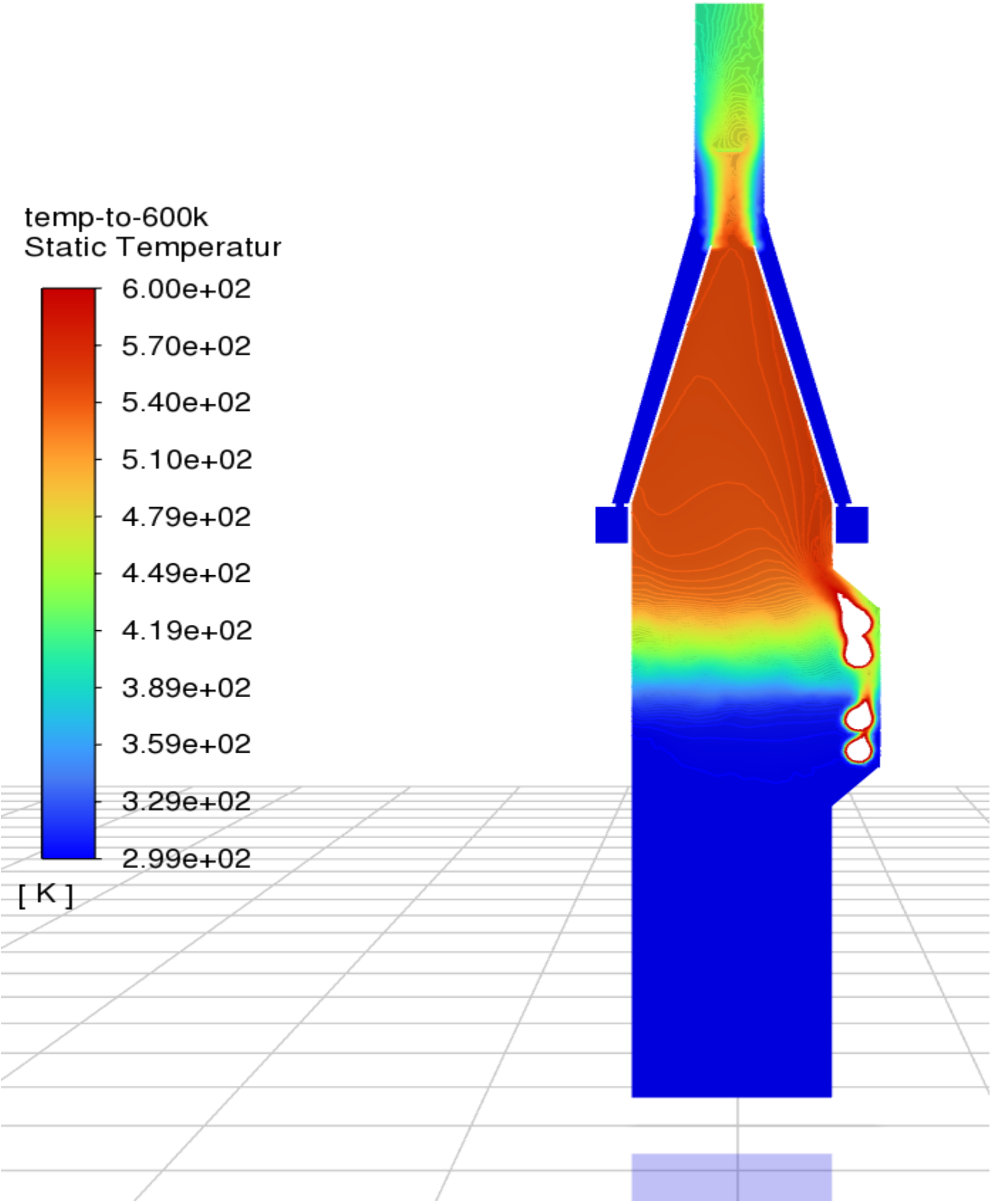


Flow through OSU calorimeter with perforated plates and baffle at exit

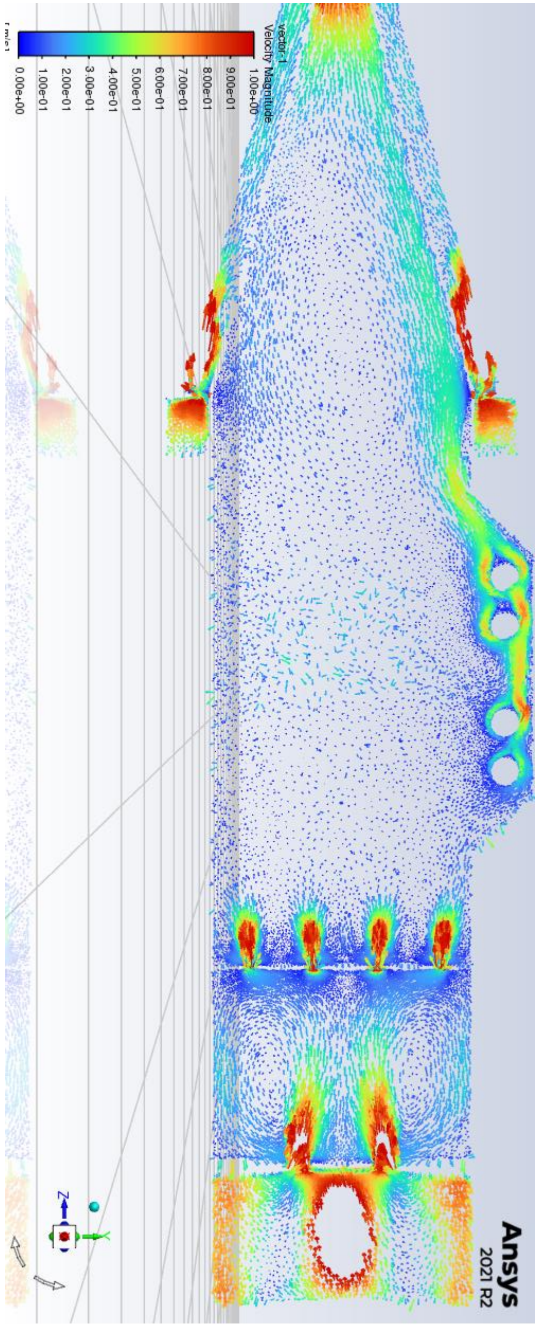
Predicted velocity field
(instantaneous view)



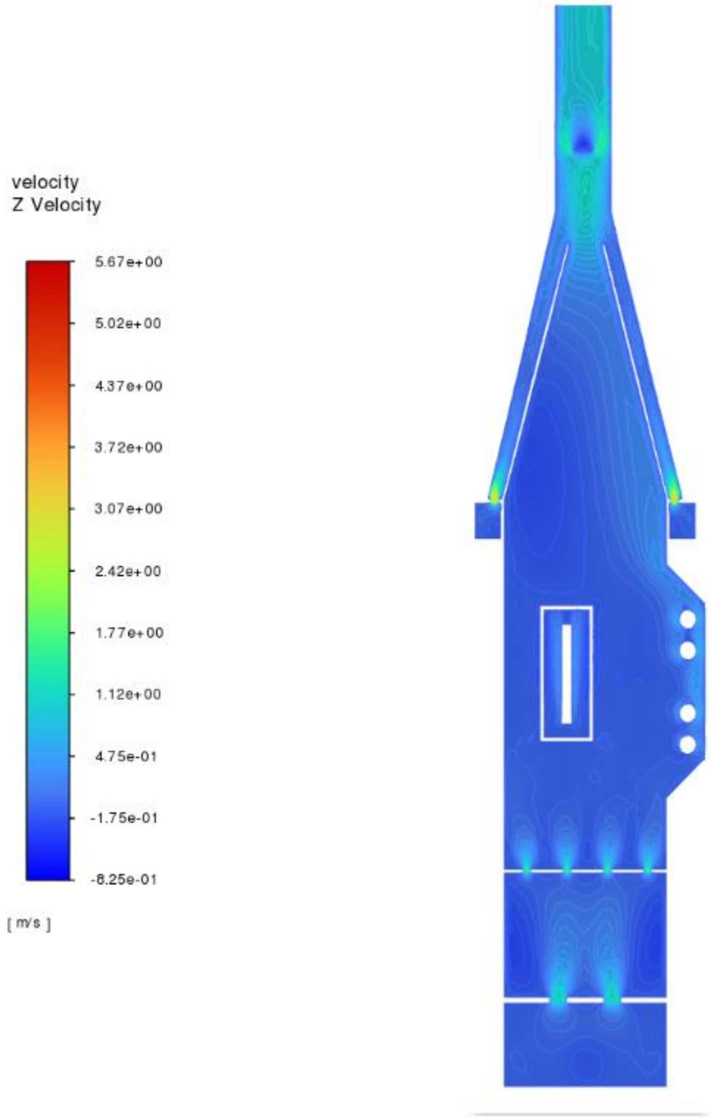
Predicted temperature field (instantaneous view)

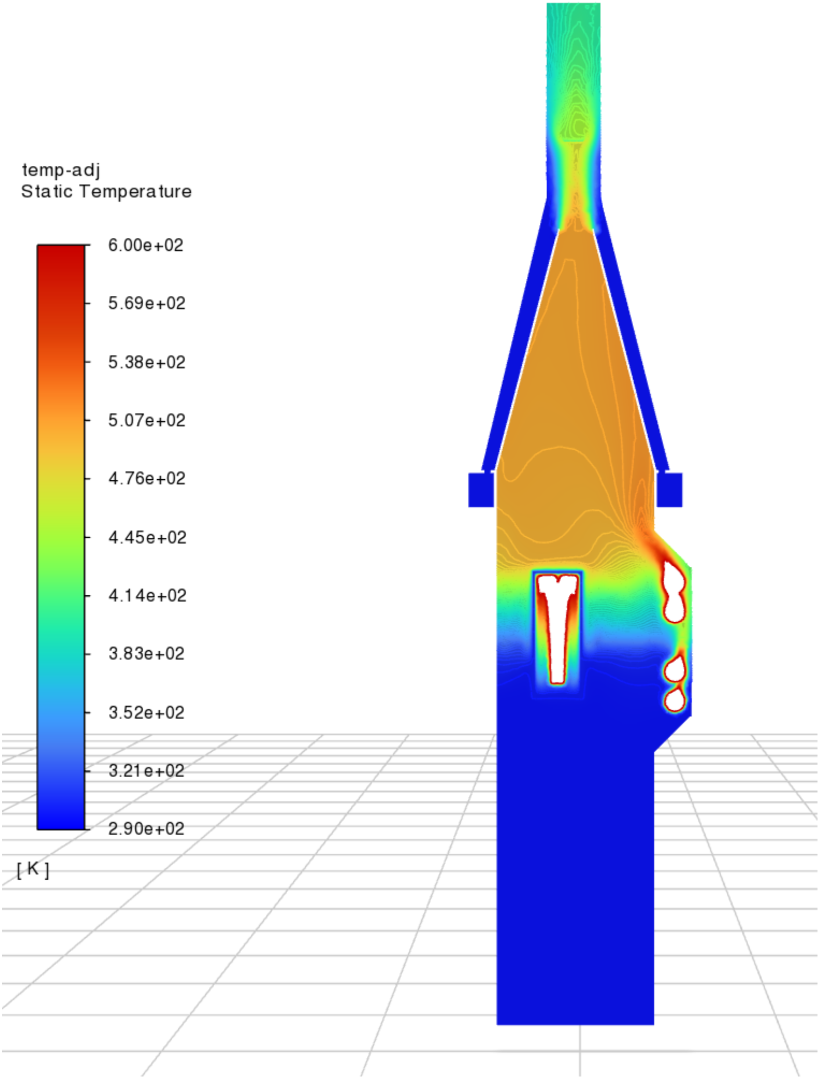


Instantaneous velocity vectors



Instantaneous view of velocity and temperature fields
OSU calorimeter with specimen-holder and specimen





Outline of the presentation

- Background and a bit of history
- Research areas and topics
- Progress to date, October 2022
- Concluding remarks

Tasks in-progress

- ANSYS-FLUENT models for the 3D transient flow and heat transfer in the OSU reactor (full-scale)

- Proper characterization of boundary conditions of all surfaces

  - insulated walls

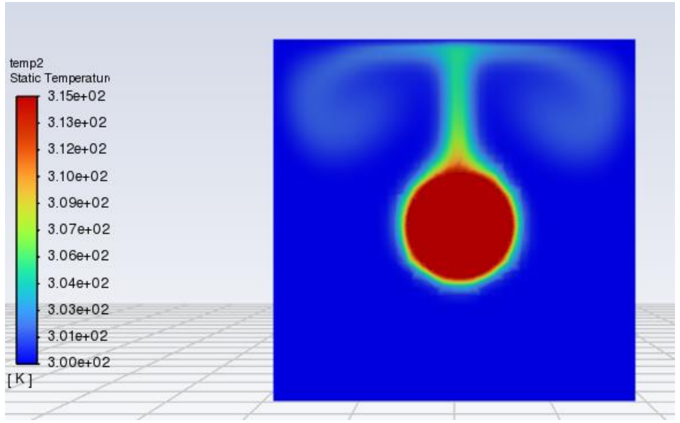
  - radiative heat flux boundary conditions for the heating elements

  - incorporate surface radiation in the present model

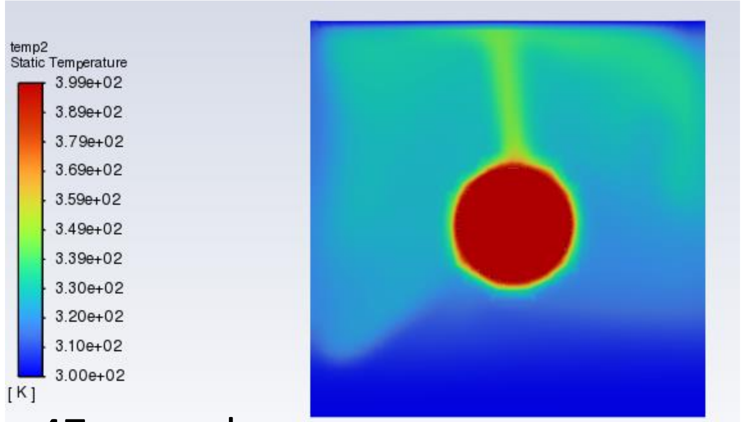
Implementation of surface radiation modeling in ANSYS-FLUENT

Natural Convection Flow around a Heated Cylinder in an Enclosure – effect of surface radiation

Temperature fields (without surface radiation)

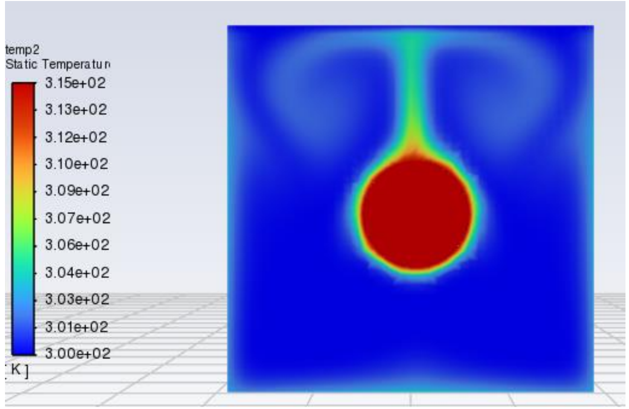


7 seconds

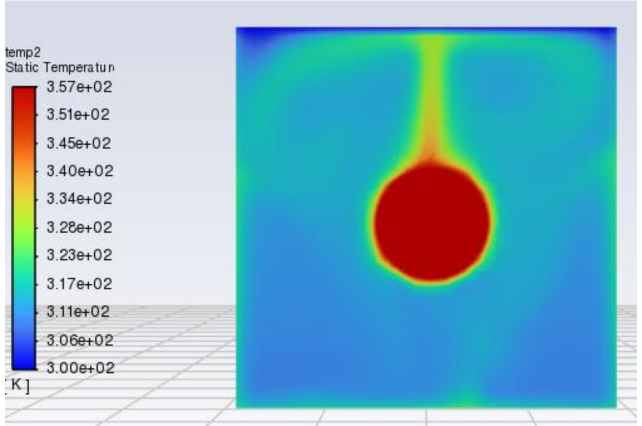


47 seconds

Temperature fields (with surface radiation)



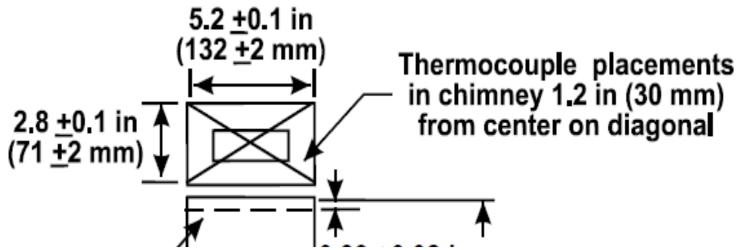
7 seconds



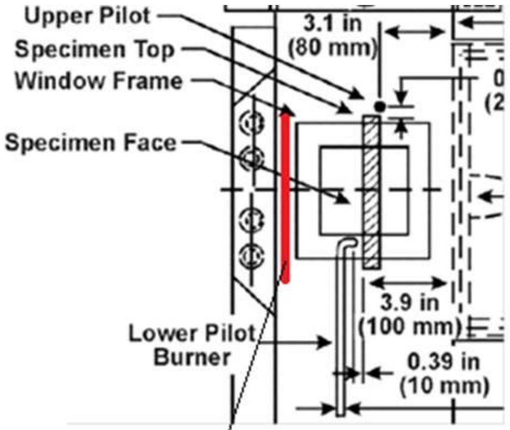
47 seconds

Solid sample combustion modeling in the OSU reactor

Solid fuel → gasification volatilization → combustion



What approach should be followed?



Diamond-shaped Mask (side view)

Specimen container

Sample specimen

Tasks to be completed……………

- Debugging and trouble-shooting the ANSYS-FLUENT model for flow and heat transfer in the OSU reactor **without any sample**
- Development of the ANSYS-FLUENT model for flow and heat transfer in the OSU reactor with **idealized 'heat release' sample**

Sub-models need to be developed for the volatilization of the samples and eventual combustion aided by the pilot flames ?

Specimen container

Sample specimen

CONCLUDNG THOUGHTS

- Developing a CFD model for the time-dependent three-dimensional convective-radiative flow, temperature and species concentration fields in the full-scale OSU reactor with sample burning is a challenge
- Considerable geometrical complexities in the reactor to facilitate mixing and heat release
- Burning of the sample in a closed container will require careful modeling considerations

Future Collaborations with FAA Tech. Center

- ANSYS-FLUENT model development for the OSU Rate of Heat Release reactor
- ANSYS-FLUENT models for the volatilization of the solid sample
- ANSYS-FLUENT model development for the OSU Rate of Heat Release reactor with sample volatilization and combustion
- Exploration of other research areas within the Fire Research Group where ANSYS-FLUENT simulations can further understand and design