



2025

DESIGN CREDIT SUMMER PROJECT

CFD ANALYSIS REPORT: FLUID FLOW IN A SERPENTINE BAFFLED REACTOR

ARYAN PRASAD
B23CH1010
INDIAN INSTITUTE OF
TECHNOLOGY JODHPUR,
RAJASTHAN

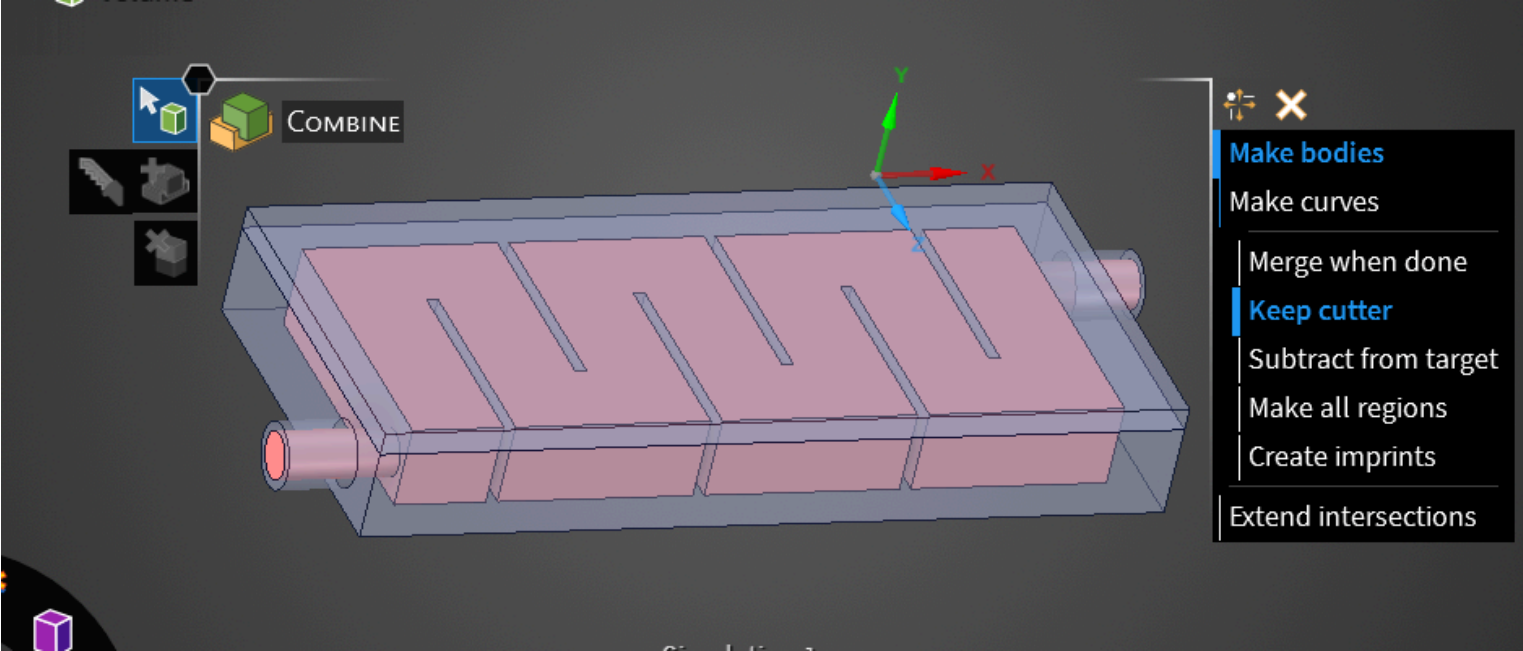


ACKNOWLEDGEMENT

I wish to express my sincere gratitude to my project supervisor, **Dr. Ramesh Asapu**, Professor in the **Department of Chemical Engineering at IIT Jodhpur**. His invaluable guidance, insightful feedback, and constant encouragement have been instrumental throughout the course of this project.

I would also like to extend my special thanks to my Teaching Assistant, **Thirumalesh B S**. His timely assistance, technical support, and patient clarification of my doubts were incredibly helpful and greatly facilitated the progress of my work.





Project Objective:

The primary objective of this simulation was to analyze the hydrodynamic characteristics of a baffled flow photoreactor. The analysis was conducted using Ansys Fluent to model the fluid behavior under a steady-state flow rate of 2 L/min. The evaluation focused on characterizing the pressure drop and velocity distribution using contour plots, streamlines, and pathlines to assess the reactor's performance and identify areas for potential design optimization.

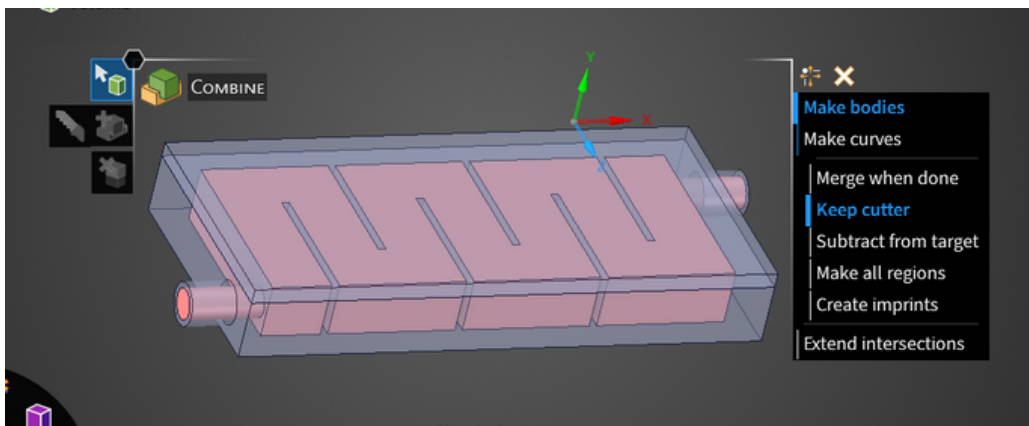
01

Pre-Processing: Geometry and Mesh Generation

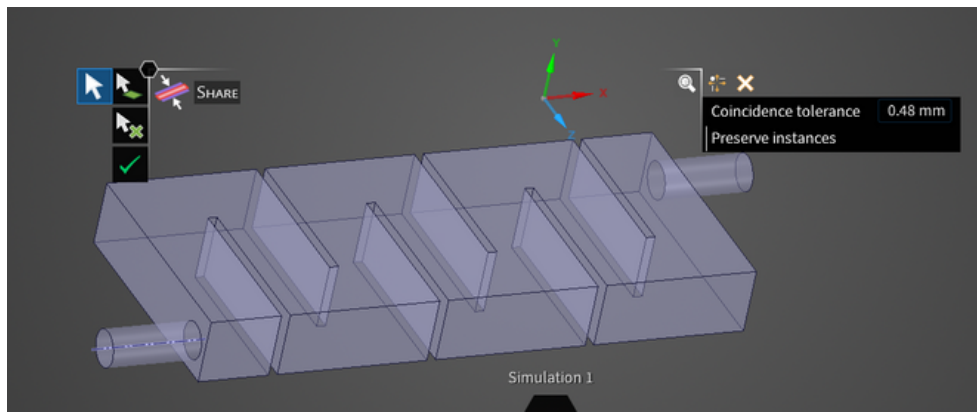
1.1. Geometry Modeling

The geometry represents a channel reactor with an inlet on one end and an outlet on the other. Internal baffles are arranged to force the fluid through a long, winding, serpentine path.

- **Software:** The geometry was prepared in a Solidworks environment, as suggested by the user interface in Model 1 .jpg.



- **Methodology:** A fluid domain was created, likely by defining a solid block for the reactor's internal volume and then using a boolean "Subtract" operation to remove the solid baffle volumes, resulting in a single continuous volume for the fluid flow (Model 1 .jpg).

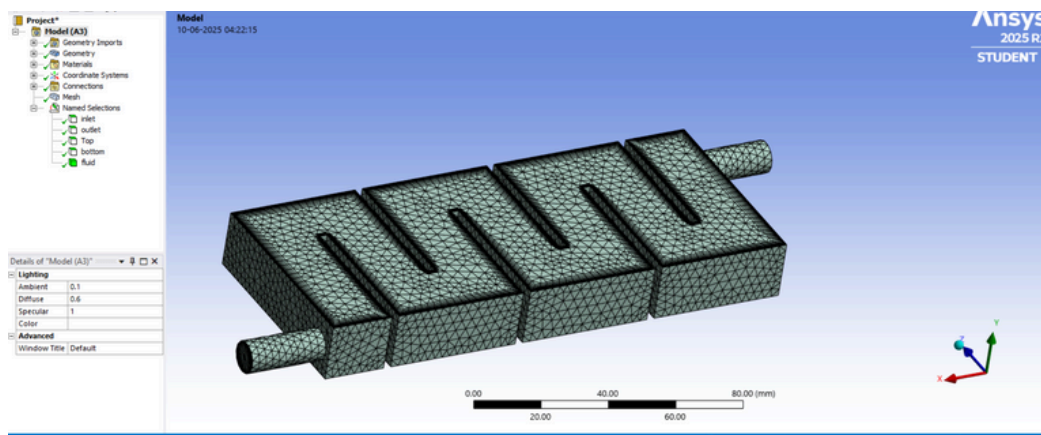


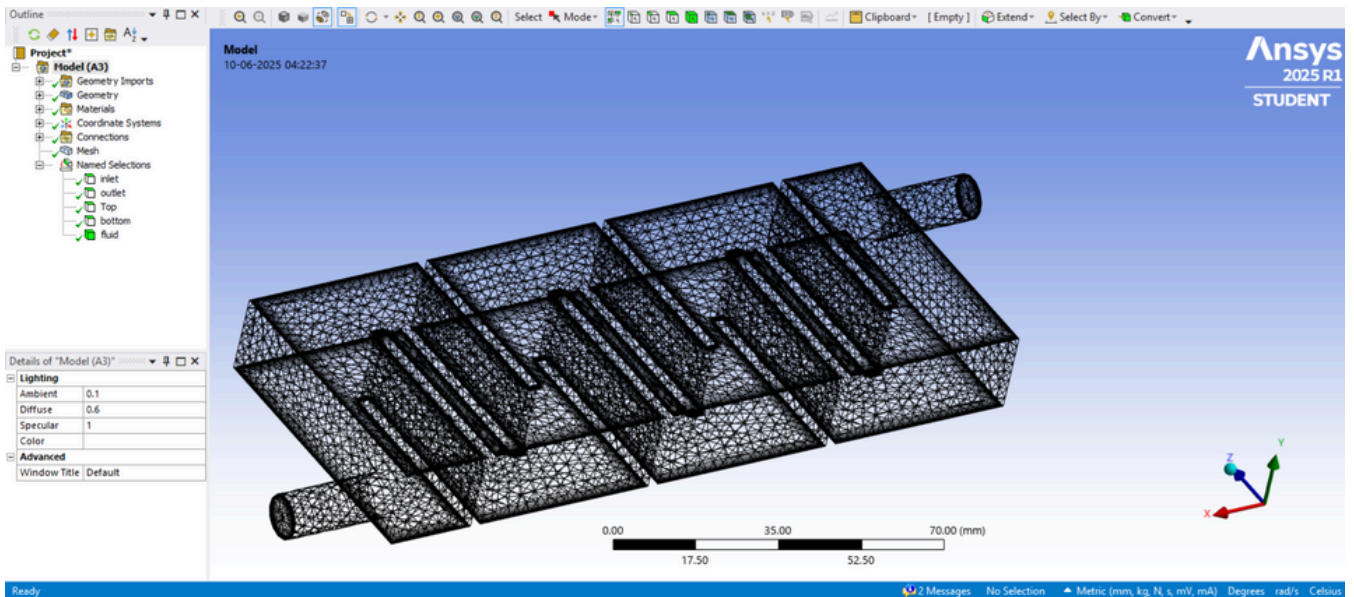
- **Topology:** The "Share" topology feature was likely used (Reactor's Volume after getting merged.jpg) to ensure that the interfaces between different conceptual parts of the volume were connected, allowing for a conformal mesh.

1.2. Mesh Generation

A computational mesh was generated to discretize the fluid domain for the numerical solver.

- **Software:** Ansys Meshing.
- **Mesh Type:** An unstructured tetrahedral mesh was employed throughout the fluid volume. This is evident from the surface mesh composed of triangular elements (New Volume Model - Meshed.jpg) and the internal volume elements (New Volume Model - Meshed (2).jpg).



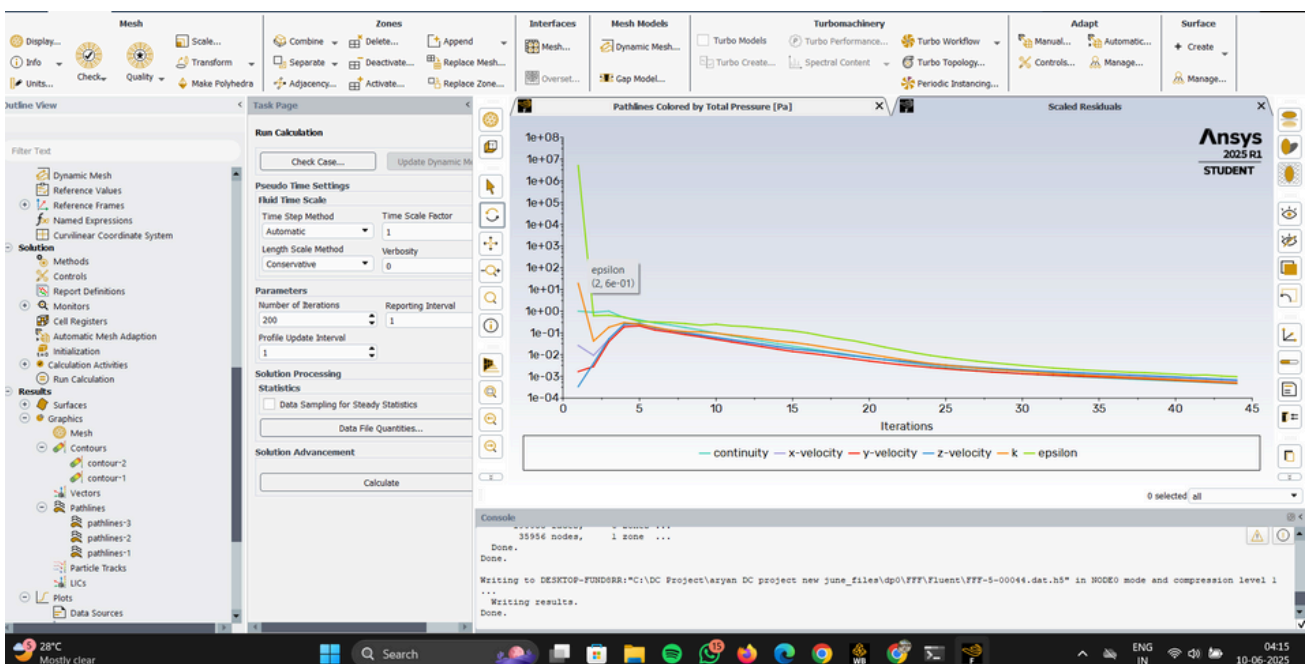


- **Named Selections:** Critical surfaces were defined with named selections for the application of boundary conditions: **inlet**, **outlet**, **Top**, **bottom**, and a **body selection for the fluid domain**.
- **Initial Assessment:** Visually, the mesh appears to be of reasonable quality, with elements distributed relatively uniformly across the domain. A formal mesh independence study would be required to ensure the results are not dependent on the mesh resolution.

Solver Setup: Ansys Fluent

Based on the user interface and results plots, the analysis was performed using the Ansys Fluent solver.

- **Analysis Type:** Steady-state, as indicated by the iterative solution process shown in the residuals plot.
- **Turbulence Model:** A two-equation turbulence model was used, specifically the **k-epsilon ($k-\epsilon$) model**. This is confirmed by the monitoring of residuals for continuity, three velocity components, turbulent kinetic energy (k), and turbulent dissipation rate (ϵ) as seen in Scaled Residuals graph.jpg. This model is a standard choice for fully turbulent internal flows.



Boundary Conditions (Inferred):

- Inlet: A velocity or mass flow rate was specified at the inlet surface.
- Outlet: A pressure-outlet boundary condition was likely applied at the outlet, probably with a gauge pressure of 0 Pa.
- Walls: A no-slip shear condition was applied to all wall boundaries (Top, bottom, and the baffle faces), where the fluid velocity is zero relative to the wall.

03

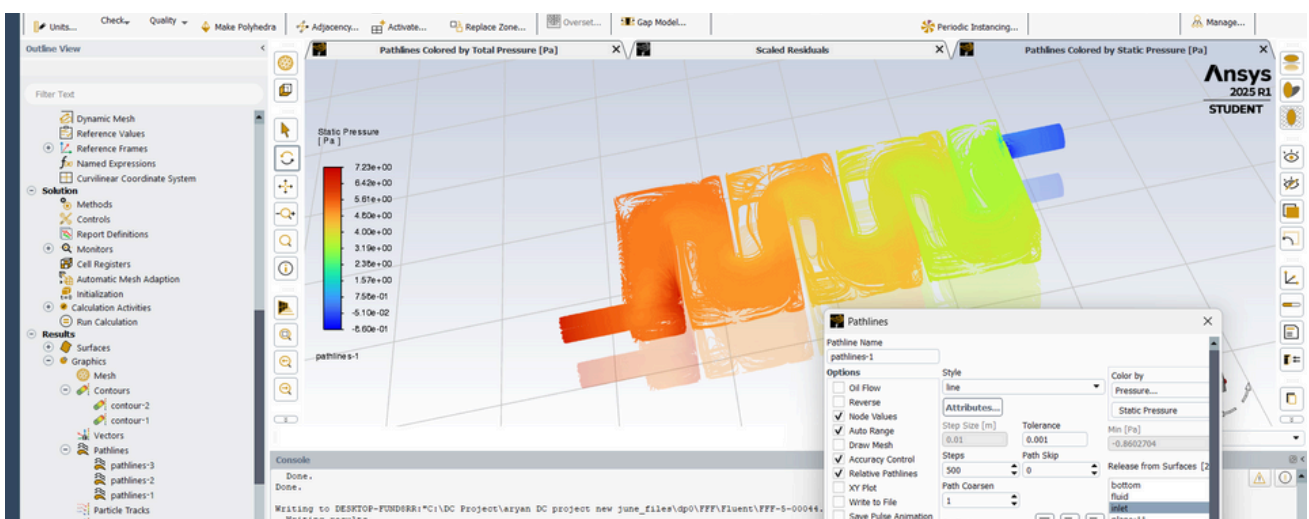
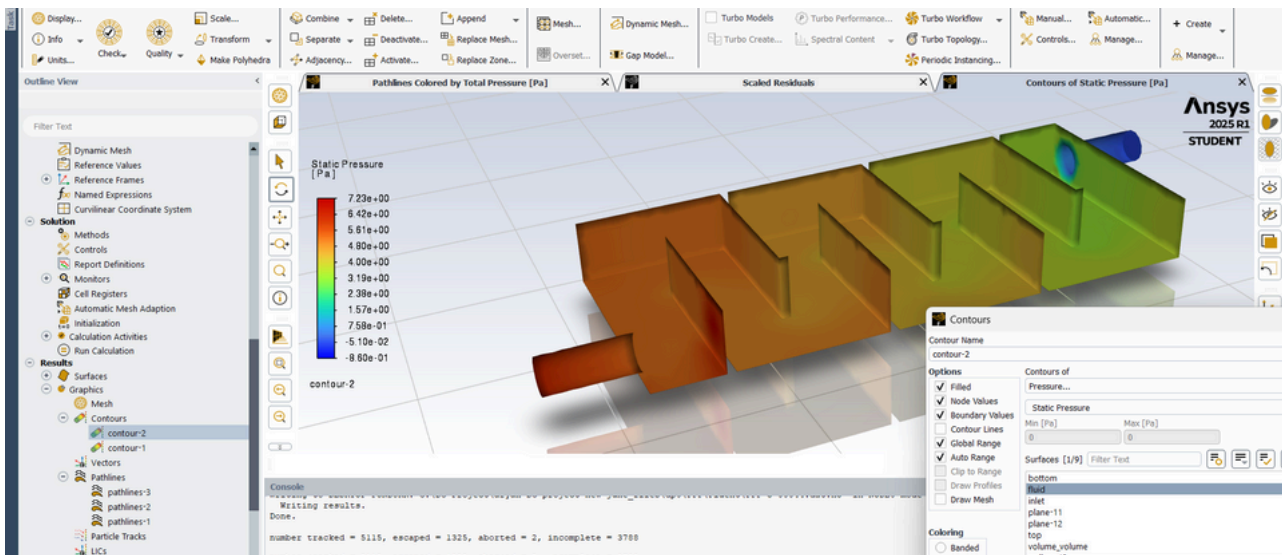
Solution Convergence

The convergence of the simulation was monitored by plotting the scaled residuals for each governing equation against the number of iterations.

- **Convergence Plot** (Scaled Residuals graph.jpg): The plot shows that all residuals dropped by several orders of magnitude and then plateaued, indicating that a stable, converged solution was reached.
- **Criteria:** The continuity, k , and ϵ residuals flattened out at a value around 10^{-4} , which is a generally acceptable convergence criterion for many industrial applications. The velocity residuals reached even lower values. The solution stabilized well before the set 200 iterations.

4.1. Pressure Distribution

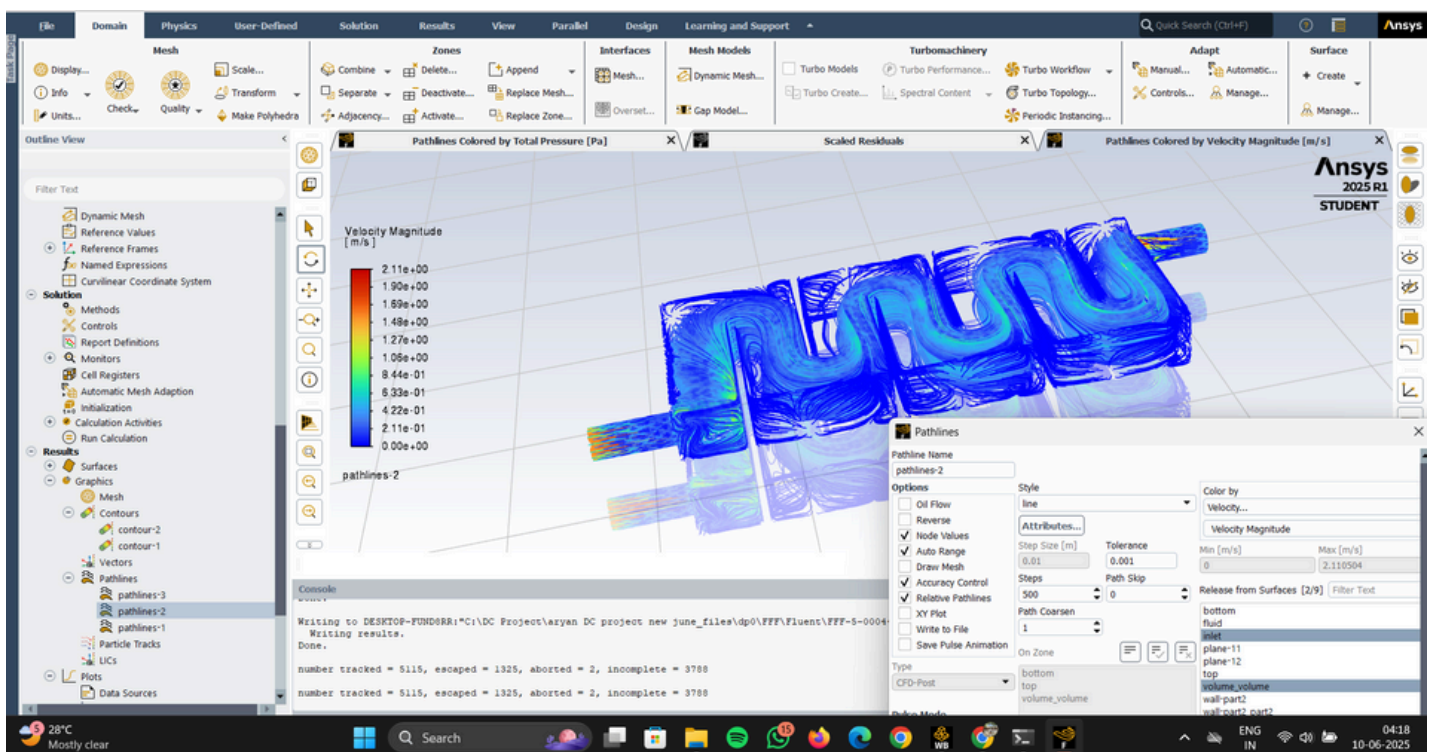
- **Observation:** A clear pressure gradient exists along the length of the reactor, as shown in the static pressure contour plot (Pressue - Pressure Magnitude Contour Plot .jpg) and the pressure-colored pathlines (Pressure Magnitude Path Lines.jpg).

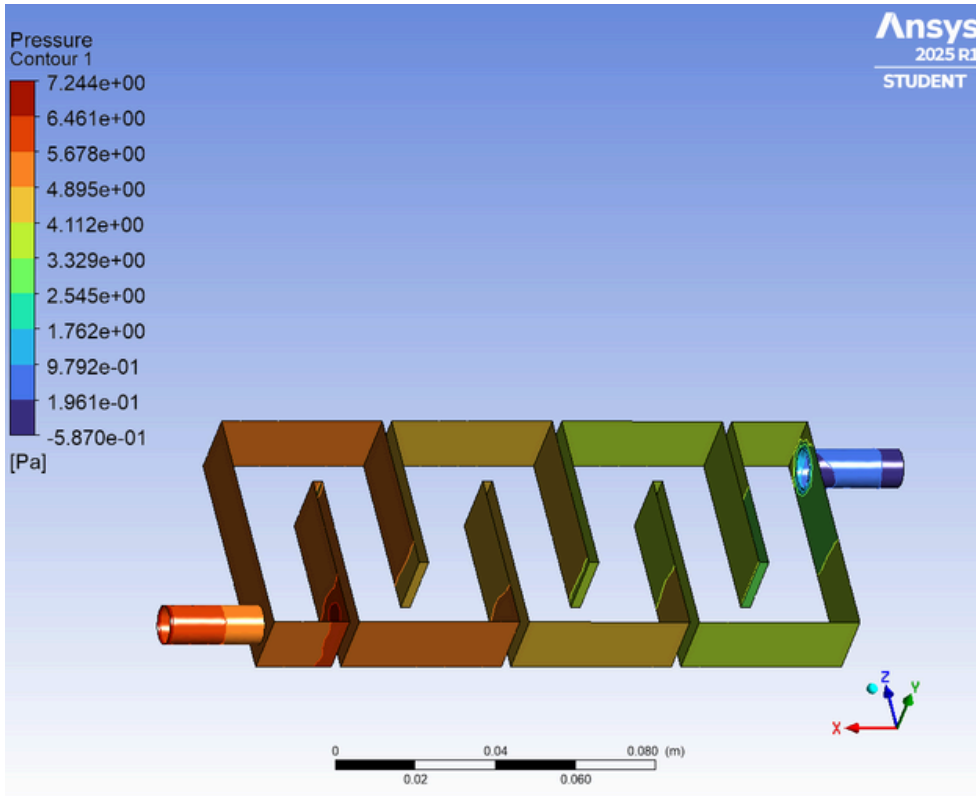


- **Analysis:** The pressure is highest at the inlet (approx. **73 Pa**, shown in green/blue) and progressively decreases through the serpentine path to its lowest point at the outlet (near 0 Pa, shown in orange/red). This pressure drop of approximately 73 Pa is the energy required to drive the fluid through the channel against viscous and frictional forces, as well as losses incurred from the multiple 180-degree turns. The pressure drop appears to be relatively uniform in each straight section of the channel.

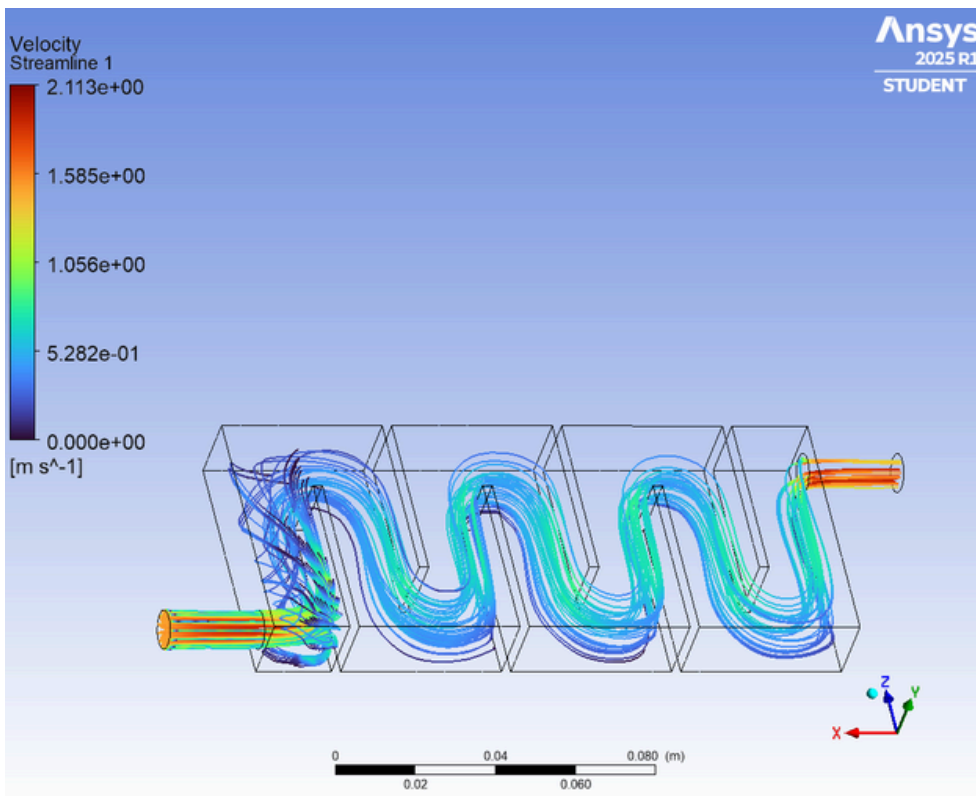
4.2. Velocity Distribution

- **Flow Path** (Velocity Magnitude Path Lines.jpg): The pathlines clearly illustrate that the fluid follows the intended serpentine path defined by the baffles. The flow is well-guided through the channels.





**Pressure
Contour 1**



**Velocity
Streamline 1**

This CFD analysis successfully characterized the hydrodynamic behavior of the baffled photoreactor at a **2 L/min flow rate**.

Key Findings:

1. The reactor induces a predictable pressure drop of approximately **73 Pa**.
2. The velocity distribution is highly **non-uniform**, characterized by high-velocity jets and significant **recirculation (dead) zones**.
3. These dead zones are detrimental to the reactor's performance, leading to a broad residence time distribution and poor mixing, thereby reducing the overall efficiency and consistency of the photochemical process.

Recommendations:

- **Design Optimization:** To improve performance, the reactor design should be modified to minimize these dead zones. Rounding the inner and outer corners of the turns or adding fillets could help streamline the flow and promote better mixing.
- **Further Analysis:** A transient simulation could be performed to study the stability of these recirculation zones and to calculate the Residence Time Distribution (RTD) curve more accurately, providing a quantitative measure of the flow non-uniformity.