

Fluid Flow Simulation in a Duct

Introduction:

This document provides an overview of a fluid flow simulation conducted on a duct using Ansys Discovery and Ansys Fluent. The aim is to analyze the flow dynamics within the duct and assess the design's effectiveness.

Assumptions:

The fluid flow is assumed to be steady-state.

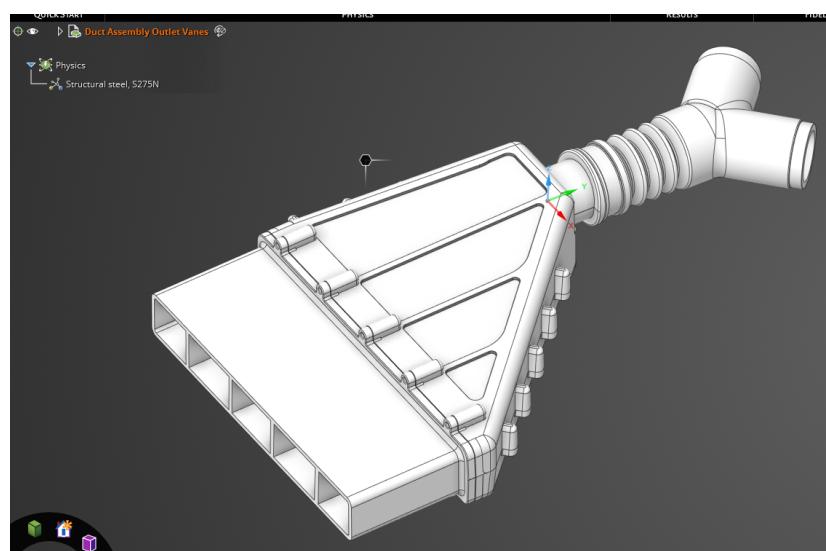
The material of the structure is steel, and air is used as the fluid.

Radiative heat transfer is considered negligible.

The properties of steel and air are taken from default properties of Ansys Discovery.

Geometry:

The geometry used in the simulation is a duct, which is shown in the below figure.



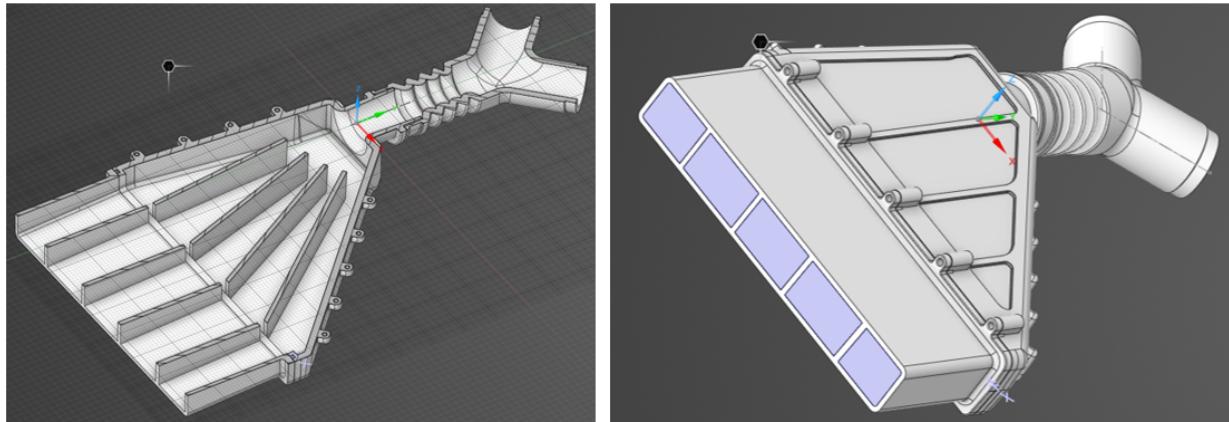
Material Data:

Structure Material: Steel

Fluid: Air

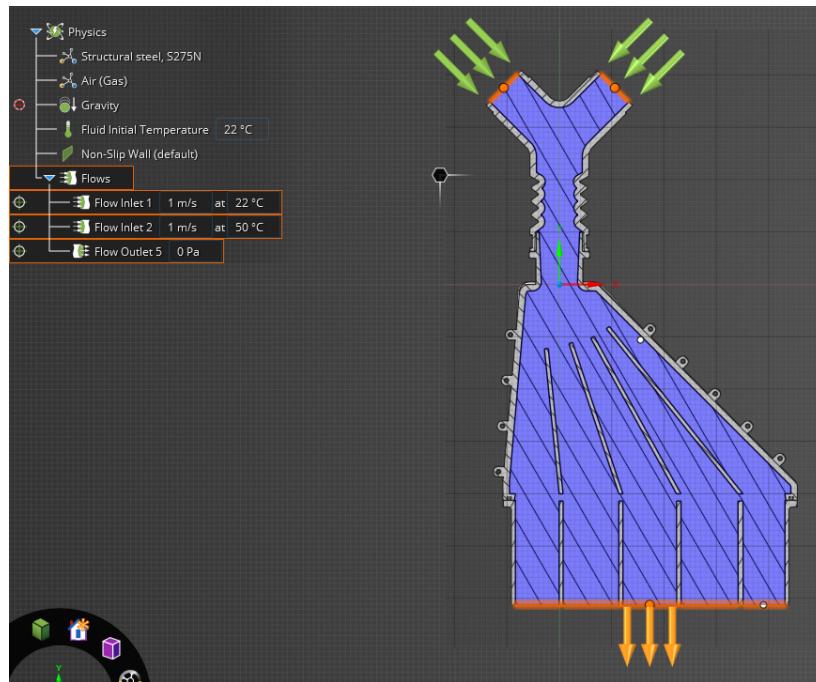
Cross-Section Analysis:

Initially, the duct's cross-section was analyzed, revealing an absence of inherent flow within the duct (as shown in the left figure below). This necessitated the manual introduction of flow volume, accomplished using the volume extract feature in the prepare tab of Discovery (illustrated in the right figure).



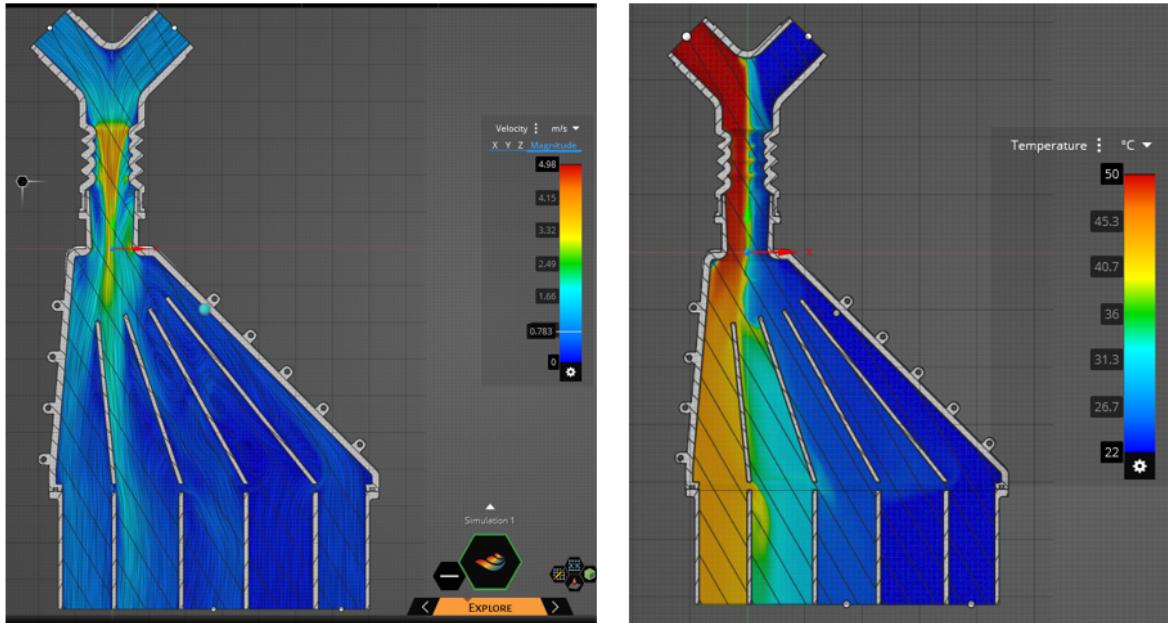
Boundary Conditions:

The boundary conditions, as detailed below, highlight a key feature: the varying temperatures at the inlets. This variation necessitates the use of Discovery to simultaneously solve for temperature and fluid velocity.

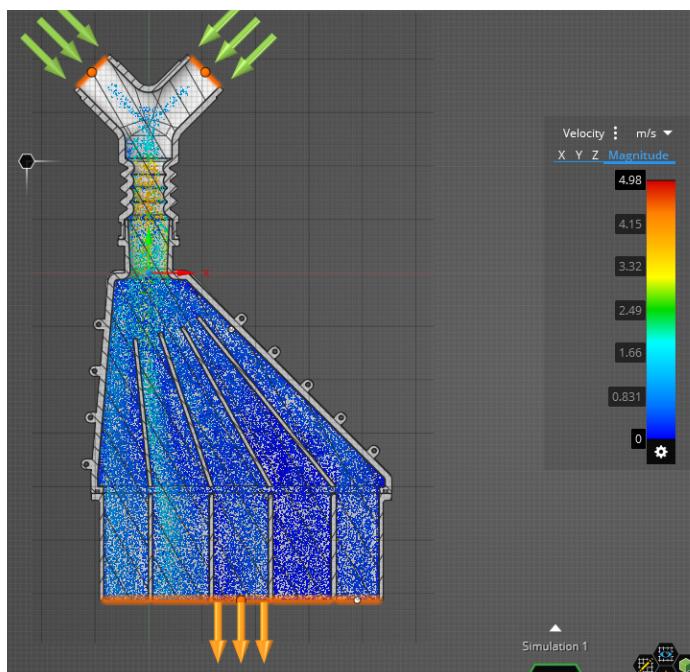


Results:

The results below illustrate the velocity streamline and temperature distribution across the duct's cross-section.

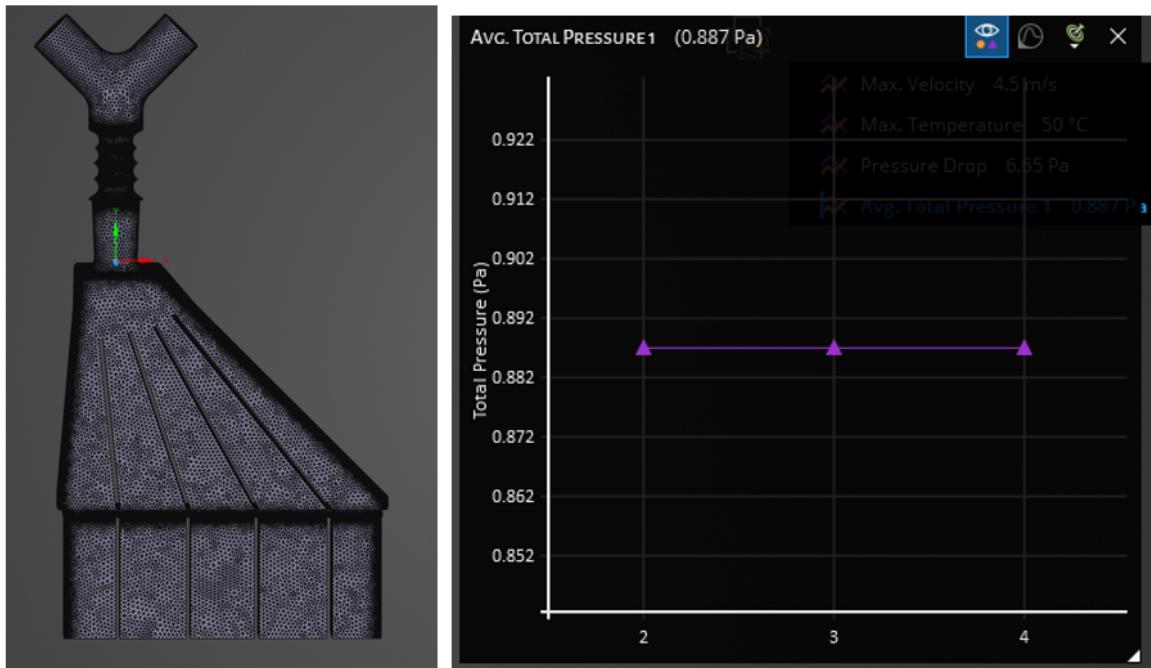


Upon analyzing the flow, it was evident that a significant portion of the flow was exiting through one outlet, suggesting a potential design inefficiency. Further examination through particle flow animation indicated that particles from other outlets were converging into the main outlet, reinforcing the notion of suboptimal design.

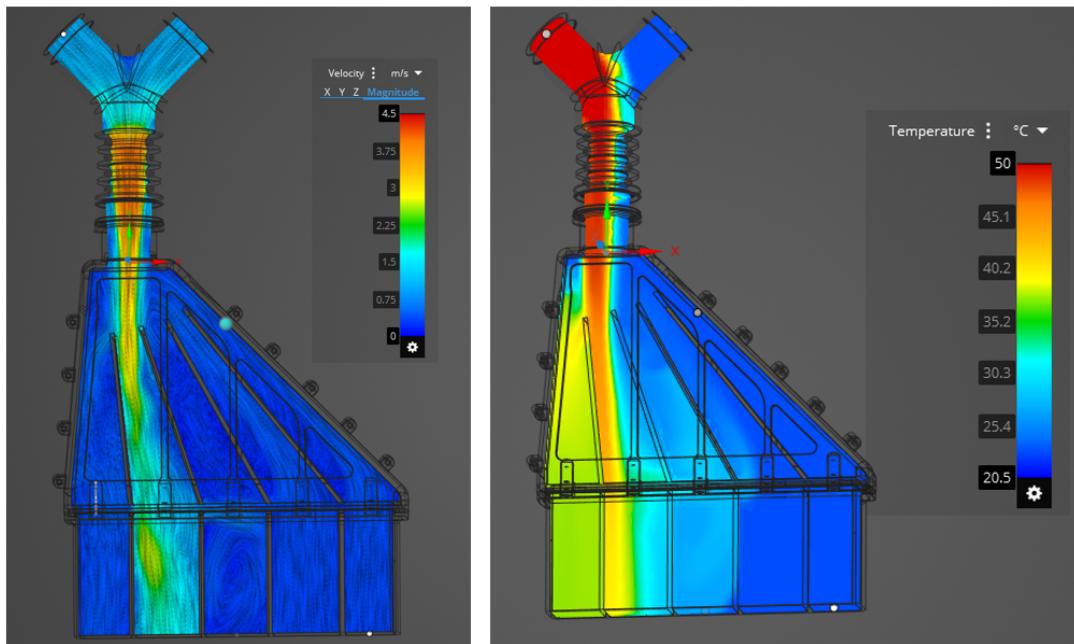


High Fidelity Analysis with Fluent:

To improve the simulation's precision, a high-fidelity analysis was performed using Ansys Fluent. The simulation settings were refined to incorporate extra fluid flow options, emphasizing the monitoring of value convergence. A mesh size of 5 mm was chosen, as depicted in the left figure below, while the total pressure was closely monitored for convergence, as illustrated in the right figure below.



The results presented below display the velocity streamline and temperature profile within the duct's cross-section.



Conclusion:

This report compares the results obtained from GPU-based and CPU-based solvers. The GPU-based solver achieved a maximum flow velocity of 4.98 m/s, while the CPU-based solver recorded 4.5 m/s. In terms of temperature, the GPU solver reported a minimum of 22°C, compared to 20.5°C for the CPU solver. The selection between GPU and CPU solvers should consider their speed advantage and the specific requirements for accuracy or speed in the simulation. The GPU solver demonstrated a 10.7% discrepancy in maximum flow velocity and a 7.3% variance in minimum temperature relative to the CPU-based outcomes.

Furthermore, the simulation identified opportunities to refine the duct design for enhanced flow distribution and efficiency. The in-depth analysis offered critical insights into the flow dynamics, guiding potential enhancements to the duct's design.

Reference and CAD file:

<https://courses.ansys.com/>