

Fluid Flow Simulation in a Manifold

Objective:

This report outlines the process and results of simulating internal fluid flow in a manifold. The objective is to demonstrate the workflow for geometry, setting up physics, obtaining solutions, and visualizing results using both GPU-powered and CPU-based solvers.

Assumptions:

The fluid flow is assumed to be steady-state.

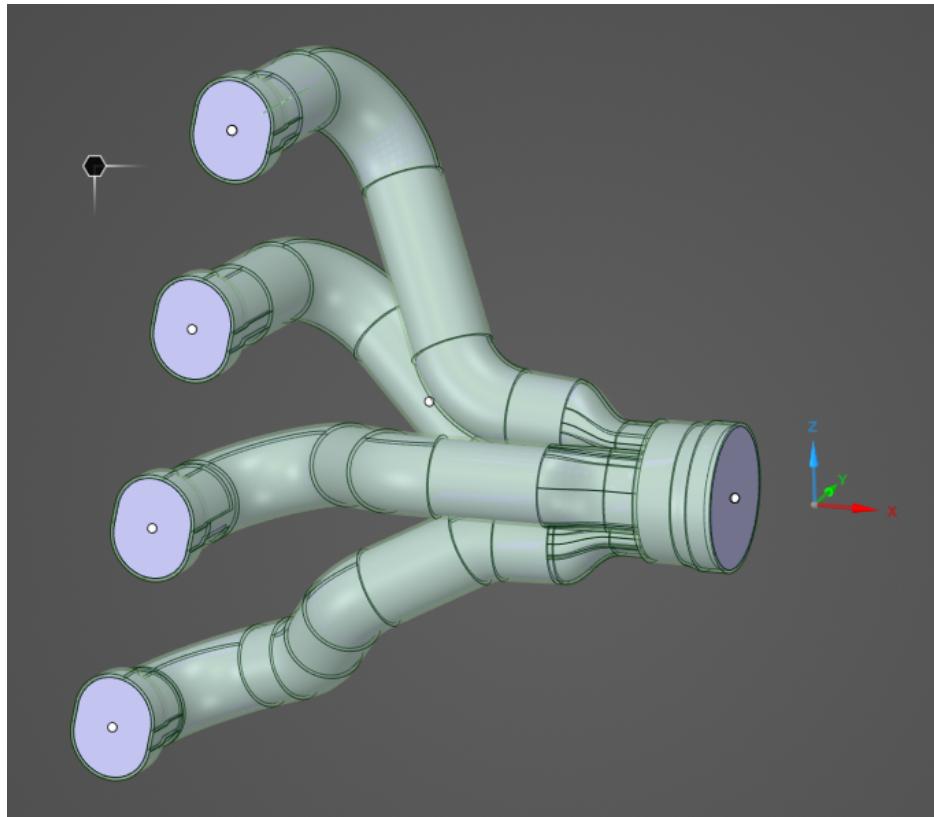
The material of the structure is steel, and air is used as the fluid.

Radiative heat transfer is considered negligible.

The properties of steel and air are assumed to be constant.

Geometry:

The geometry used in the simulation is a manifold with specific dimensions, which are shown in the below figure.



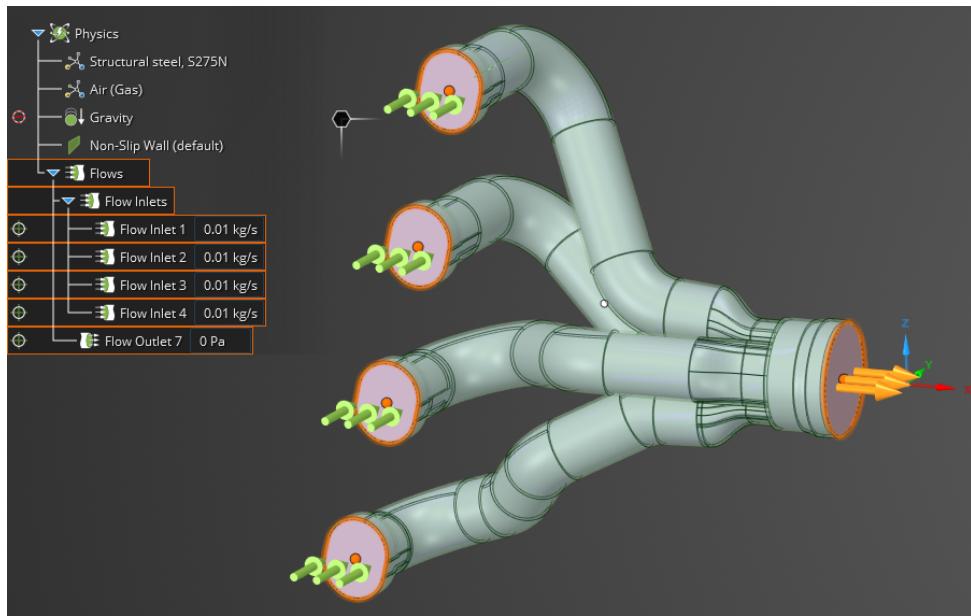
Material Data:

Structure Material: Steel

Fluid: Air

Boundary Conditions:

The boundary conditions for the simulation are as follows:



Simulation Process:

GPU-powered Solver (Ansys Discovery):

The simulation is first performed using Ansys Discovery, focusing on the GPU-powered solver.

CPU-based Solver (Ansys Fluent):

The simulation is then conducted using Ansys Fluent, emphasizing the CPU-based solver.

A refine mode is utilized, and a new session is started from scratch for this part.

The simulation with Ansys Fluent provides additional accuracy, fidelity, control, and inputs. However, running speed gets lower.

A mesh size of 5 mm is used on the body, with automatic mesh refinement in areas of curvature.

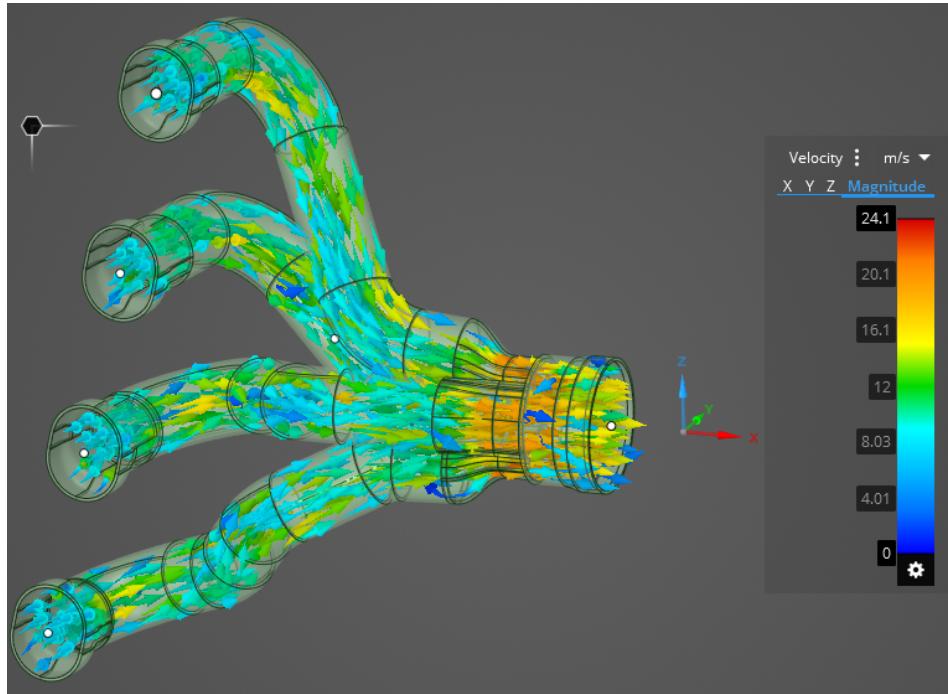
Convergence is monitored using average static pressure and velocity vectors.

The results section would detail the findings from both the GPU-powered and CPU-based solver simulations, including flow patterns, velocity distributions, and any other relevant outcomes.

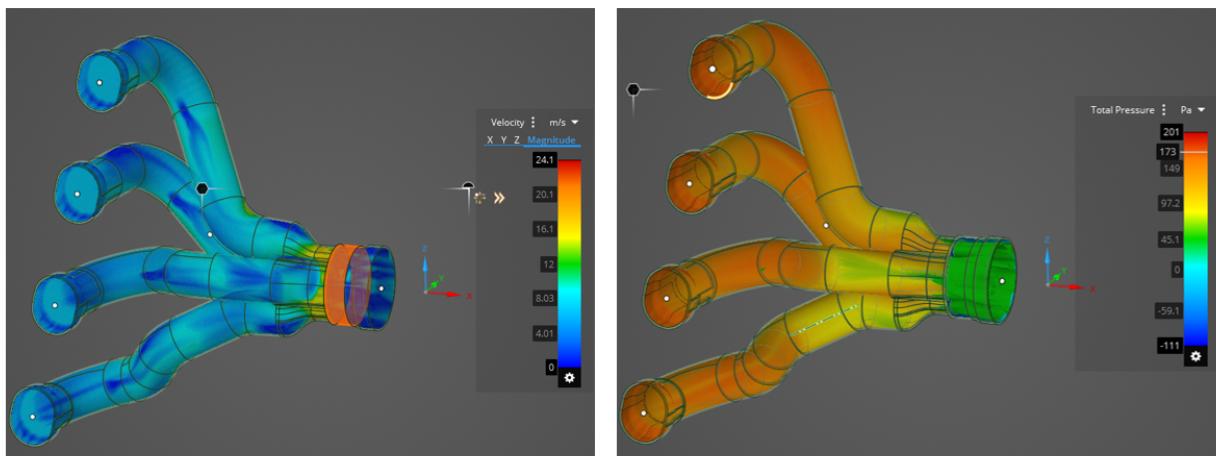
Results:

GPU-powered Solver (Ansys Discovery) results:

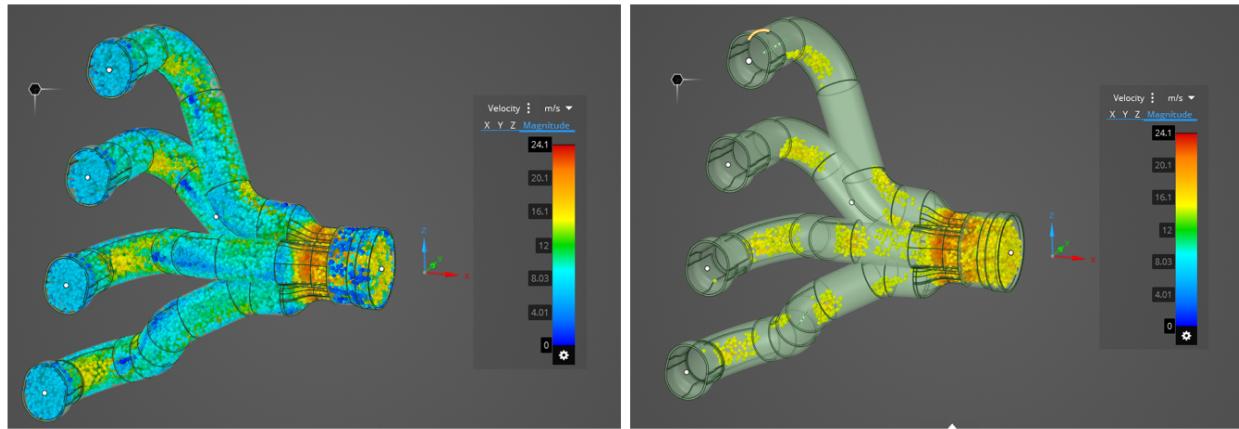
Results and visualizations are obtained for the internal flow. Below figure shows the plot for velocity vector.



Additionally, the following plots depict contour maps of flow velocity and pressure:



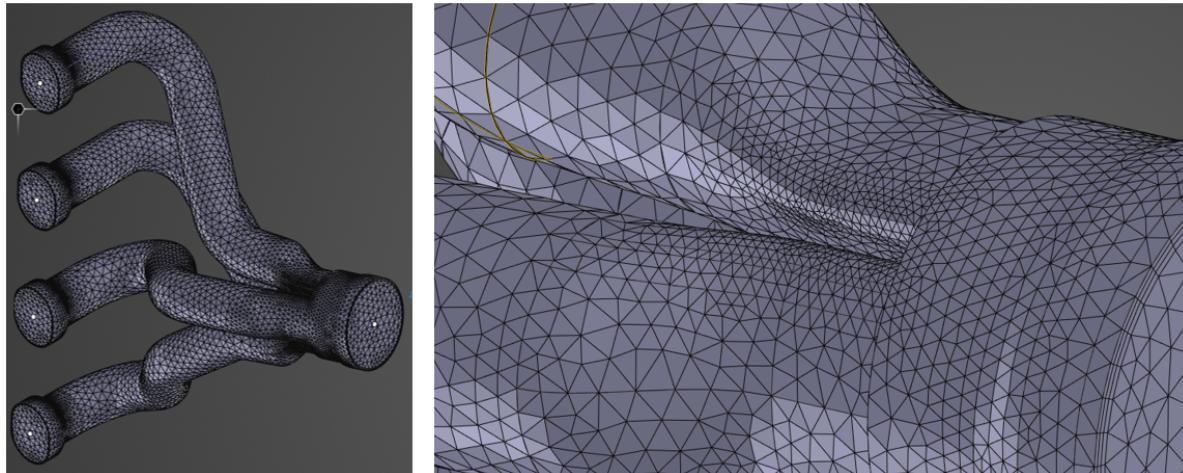
Additionally, the image below displays the particle velocities (left plot) and highlights the particles with the highest velocity (right plot).



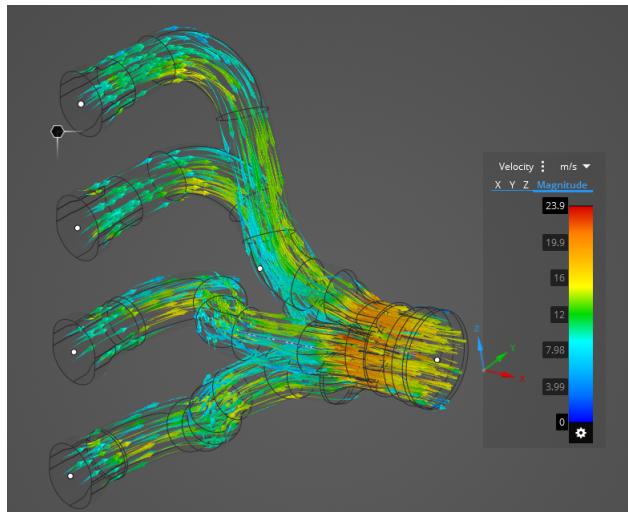
Specific results, such as the particles with the highest velocity and convergence plots, are mentioned but not detailed in the provided content.

CPU-based Solver (Ansys Fluent) results:

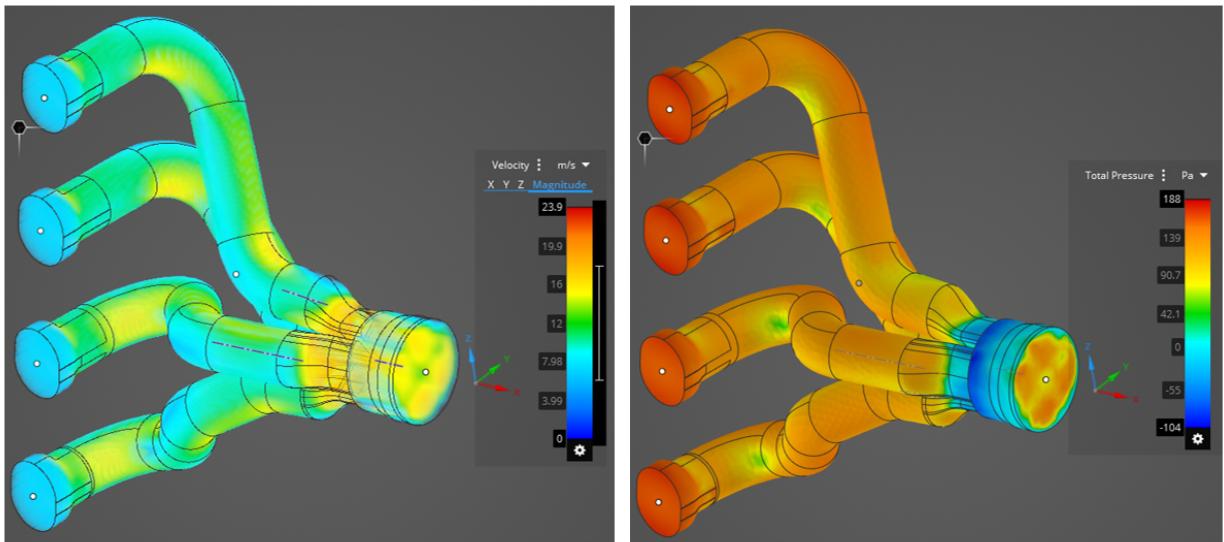
Here, we use 5 mm mesh on the body. We can see that it has automatic mesh refinement in areas of curvature, as shown the right figure below:



We assess convergence by monitoring the average static pressure, with the solver delivering the final result once convergence is attained following necessary iterations. The figure below illustrates the velocity vectors:



Additionally, the following plots depict contour maps of flow velocity and pressure:



Conclusion:

This report evaluates the outcomes from GPU-based and CPU-based solvers. The GPU-based solver shows a maximum flow velocity of 24.1 m/s, compared to 23.9 m/s with the CPU-based solver. Additionally, the total pressure is 201 Pa for the GPU-based solver and 188 Pa for the CPU-based solver. Considering the GPU solver's speed advantage, the choice between GPU and CPU solvers hinges on the need for accuracy versus speed in simulations. Notably, the GPU solver exhibits a 0.8% error in maximum flow velocity and a 6.9% error in total pressure compared to the CPU-based results.

Reference and CAD file:

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