

Setup:

The initial geometry was produced using the engineering schematics provided in the project assignment document using the ANSYS DesignModeler application taking advantage of system symmetry.

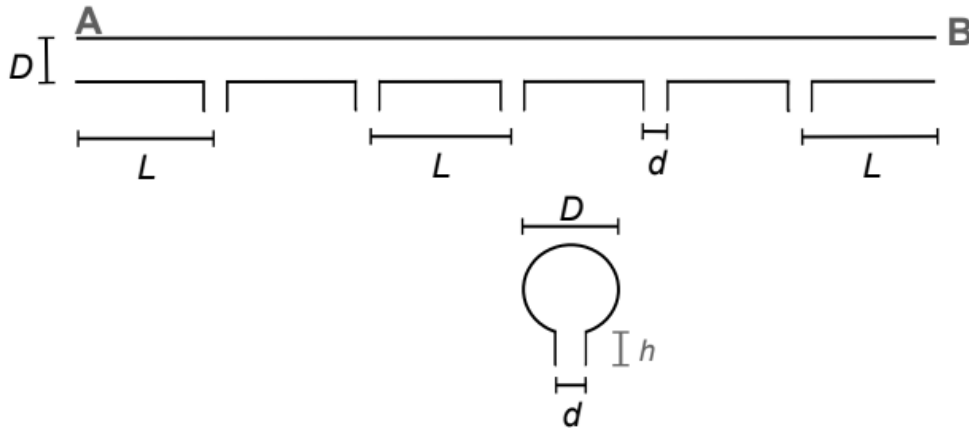


Fig. 0.1 – Provided Geometry

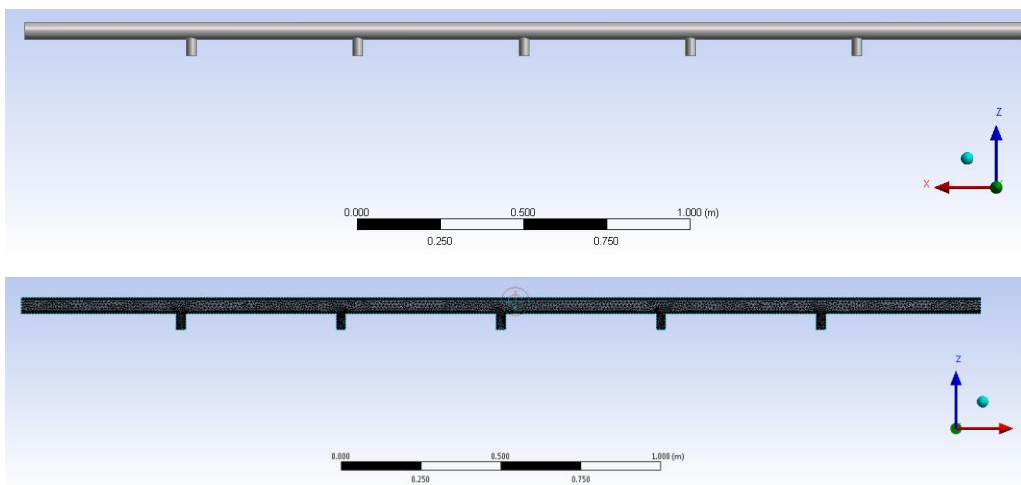


Fig. 0.2 – ANSYS 3D Model and Mesh

The resulting model was then meshed using the ANSYS Meshing application. Named sections were created for the plane of symmetry, the velocity defined inlet, and the outflow defined outlets. The Advanced Sizing Function was set to curvature and the Automatic Tet Inflation was set to Program Controlled consistent with previous applications.

Task 1:

For task 1, the analysis type was set to steady, the solver type set to pressure based, and viscous effects were set to realizable $k-\epsilon$. Liquid water was imported from the ANSYS material database and used throughout the simulation. All solutions were performed using second order discretization.

Case 1:

The inlet boundary condition was set to velocity inlet with a velocity of 0.5 m/sec in the x-direction. Both the small side pipes and the end of the main pipe opposite the inlet were set to pressure outlet with backflow. The system was initialized and run to convergence. The surface integral functionality was used to calculate the mass flow through each side pipe, with twice the value extracted from ANSYS being used to account for the use of a plane of symmetry model. The results of mass flow rate for each side pipe, along with plots of static pressure and x-velocity along the central axis of the main pipe, are presented below.

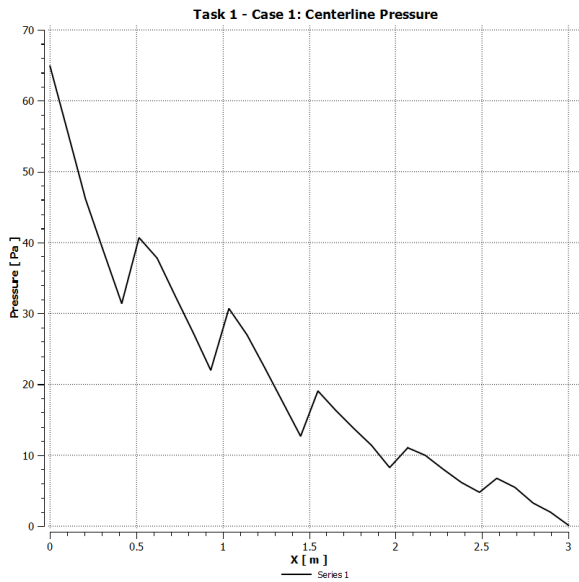


Fig. 1.1.1 – Static Pressure

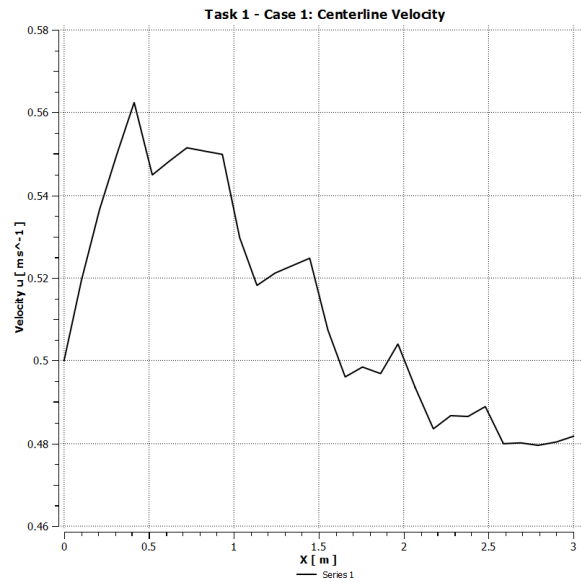


Fig. 1.1.2 – X-Velocity

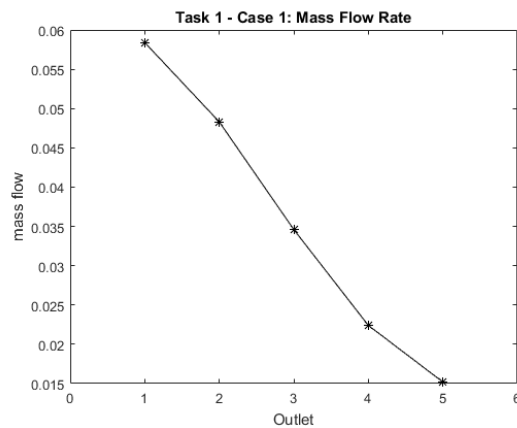


Fig. 1.1.3 – Mass Flow Rate

Case 2:

The setup and execution of Case 2 was identical to that of Case 1, with the exception that the boundary condition for the end of the main pipe opposite the inlet was changed from pressure outlet to a solid wall. The results for Case 2 are presented below.

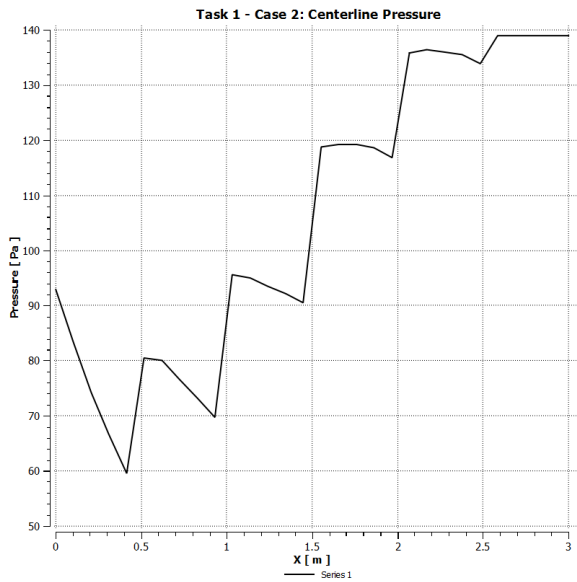


Fig. 1.2.1 – Static Pressure

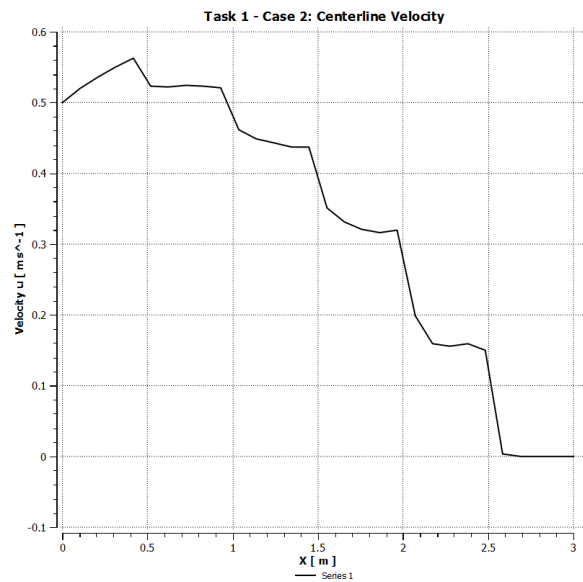


Fig. 1.2.2 – X-Velocity

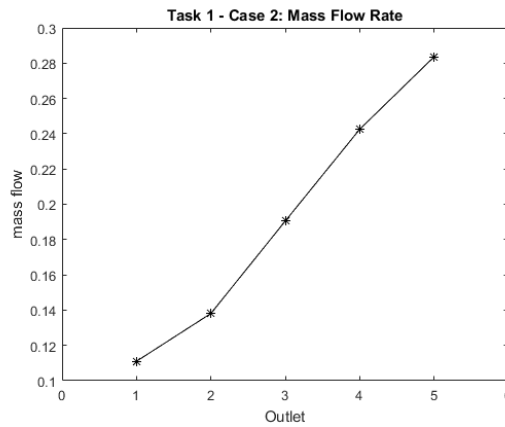


Fig. 1.2.3 – Mass Flow Rate

Task 2:

Two separate methods were attempted to achieve an approximately uniform mass flow rate. For the first, hand calculations were performed using an incompressible, inviscid model and the diameters of the side pipes and the open diameter of the end of the main pipe were varied to achieve a theoretically uniform outflow. Unfortunately, due to the viscous effects and presence of backflow, this method achieved extremely poor results.

Instead, a much simpler method was utilized to achieve uniform outflow. The various settings and boundary conditions were set to be identical to those of Task 1, Case 2, with the exception that the geometry was modified to have side pipes with a diameter of 1 cm and re-meshed as pictured below. The geometry was kept otherwise identical to that used in Task 1.

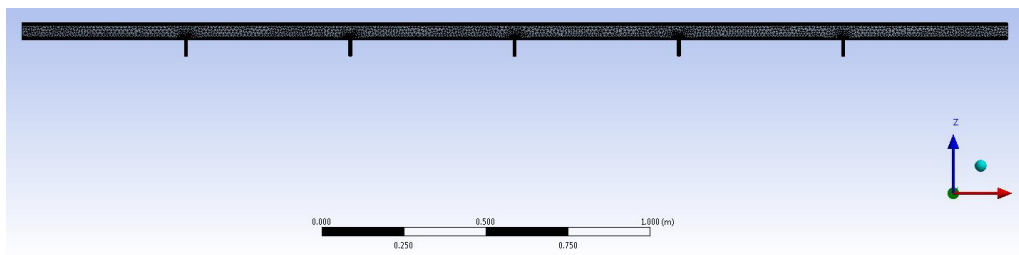


Fig. 2.1 – ANSYS 3D Mesh

The simulation was rerun, with the results pictured below.

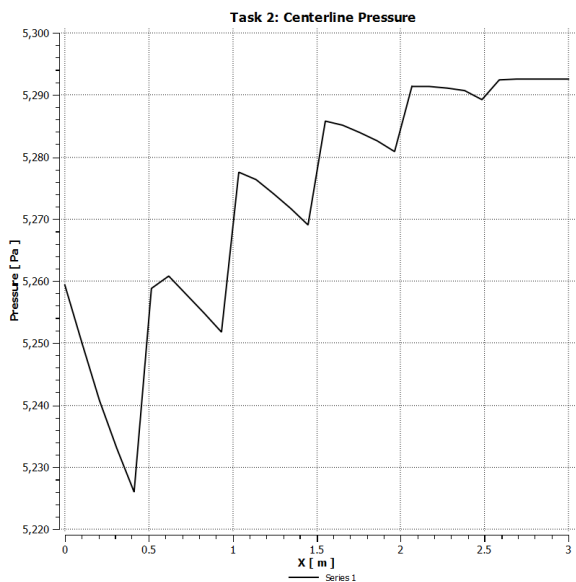


Fig. 2.2 – Pressure

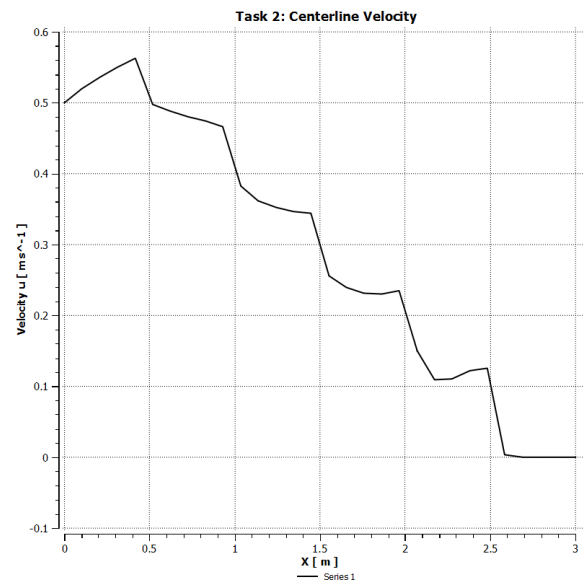


Fig. 2.3 – X-Velocity

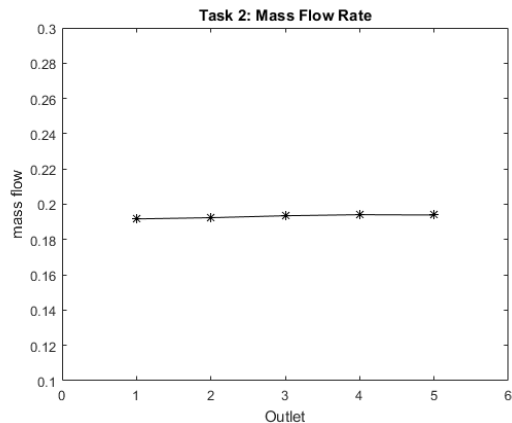


Fig. 2.4 – Mass Flow Rate

As can be seen in the above plots, the contraction of the side pipes effectively converts the main pipe into a pressure vessel driven by the constant flow rate of the velocity inlet, and since the cross section of each of the pipes is small relative to the cross section of the main pipe, the pressure drop across each is relatively small, leading to a fairly consistent mass flow rate.

In order to describe the above effect in more mathematical terms, we can introduce an error measurement given by:

$$S = \frac{1}{M} \sqrt{\frac{1}{5} \sum_{k=1}^5 (m_k - M)^2}$$

Fig. 2.5 – Error Equation

Which for the data obtained for Task 2 yields a value $S = 0.0048$, compared to a value of $S = 0.3307$ for Task 1, Case 2, which would tend to indicate an acceptable result.

Appendix A – Code

```
%Ryian Hunter
%1206137513
%MAE 598
%7 Nov. 2017

%%
%1.a.
m_dot_1 = [0.0584,0.0483,0.0346,0.0224,0.0152];
x = [1,2,3,4,5];
figure(1)
grid on
plot(x,m_dot_1,'k-*)
xlabel('Outlet')
ylabel('mass flow')
title('Task 1 - Case 1: Mass Flow Rate')
xlim([0,6])

%%
%1.b.
m_dot_2 = [0.1110,0.1381,0.1905,0.2425,0.2836];
x = [1,2,3,4,5];
figure(2)
grid on
plot(x,m_dot_2,'k-*)
xlabel('Outlet')
ylabel('mass flow')
title('Task 1 - Case 2: Mass Flow Rate')
xlim([0,6])
ylim([0.1,0.3])
M_1 = sum(m_dot_2)/5;
S_1 = (1/M_1)*sqrt((1/5)*sum((m_dot_2-M_1).^2));

%%
%2. (Control Volume Version)
m_dot_3 = [0.0595,0.0650,0.0812,0.1260,0.2418];
x = [1,2,3,4,5];
figure(3)
grid on
plot(x,m_dot_3,'k-*)
xlabel('Outlet')
ylabel('mass flow')
title('Task 2: Mass Flow Rate')
xlim([0,6])
ylim([0.1,0.3])

%%
%2. (Lazy Version/Version Used)
m_dot_4 = [0.1917,0.1924,0.1935,0.1941,0.1939];
x = [1,2,3,4,5];
figure(4)
grid on
plot(x,m_dot_4,'k-*)
xlabel('Outlet')
ylabel('mass flow')
title('Task 2: Mass Flow Rate')
xlim([0,6])
ylim([0.1,0.3])
M_2 = sum(m_dot_4)/5;
S_2 = (1/M_2)*sqrt((1/5)*sum((m_dot_4-M_2).^2));

%Ryian Hunter
```