Henry Rossiter
Daniel Fonseka
Gareth Meredith
Braeden Conrad

OpenFOAM Assignment 3

Governing Equations

$Mach = u/a$ where:

$u = flow\ velocity$

$a = speed\ of\ sound = \sqrt{\frac{\gamma p}{\rho}} = \sqrt{\frac{\gamma RT}{(Mm)}}$

$\gamma = adiabatic\ index = 1.4$

$Mm = molecular\ mass\ of\ fluid$; $R = gas\ constant$; $T = temperature\ (K)$;

$Mm = \frac{\rho RT}{p} = \frac{1.4*8314*1}{1} = 11639.6\ g/mol$

From the normal shock tables for flow with upstream $Ma = 3$, we are provided the ratios $\frac{p_1}{p_{02}} = 0.08291$ and $\frac{p_2}{p_1} = 10.333$ where $p_{02} = the\ downstream\ stagnation\ pressure$, $p_1 = the\ upstream\ freestream\ pressure$, and $p_2 = the\ downstream\ freestream\ pressure$. Since $p_1 = 1\ pa$ is known, the ratios above can be used to solve for $p_{02} = 12.06\ pa$ and $p_2 = 10.333\ pa$. Furthermore, from the normal shock tables the ratio of $p_{02}\ to\ p_{01}\ (upstream\ stagnation\ pressure)$ is equivalent to $\frac{p_{02}}{p_{01}} = 0.3283$. From this we are able to solve for $p_{01} = 36.73\ pa$ and thus complete our estimation of the stagnation pressure experienced by the normal shock at x = 0.6.

Generating a Mesh

Using OpenFOAM's blockMesh utility, we created a mesh schematic of the forward facing step. Our preliminary mesh contains:

- 16 points (8x2)
- 3 blocks
- 4 Boundary types
    - Inlet
    - Outlet
    - Obstacle
    - Bottom/Top

The mesh's global numbering scheme and boundary conditions are visualized in the schematics below.

Figure 1. Schematic of the mesh's global vertice numbering scheme (shown in 2D). For each vertice N at $z = 0$ in the schematic shown there exists another vertice $N + 8$ at $z = 0.05$. The entire set of 16 vertices defines our 3D mesh.

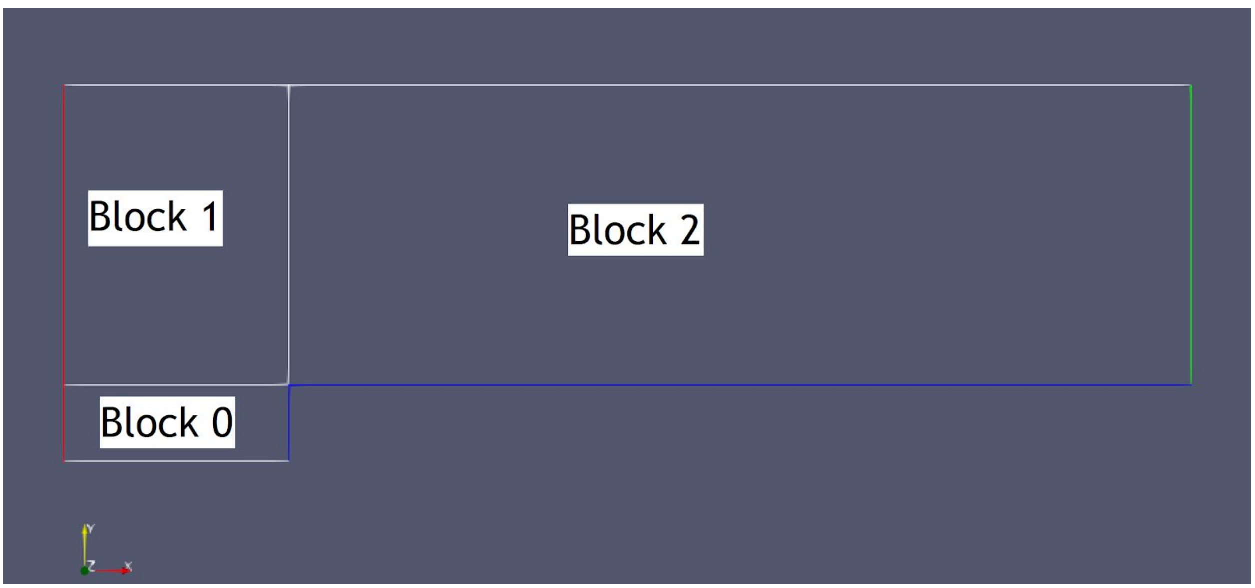


Figure 2. Schematic of block numbering scheme.

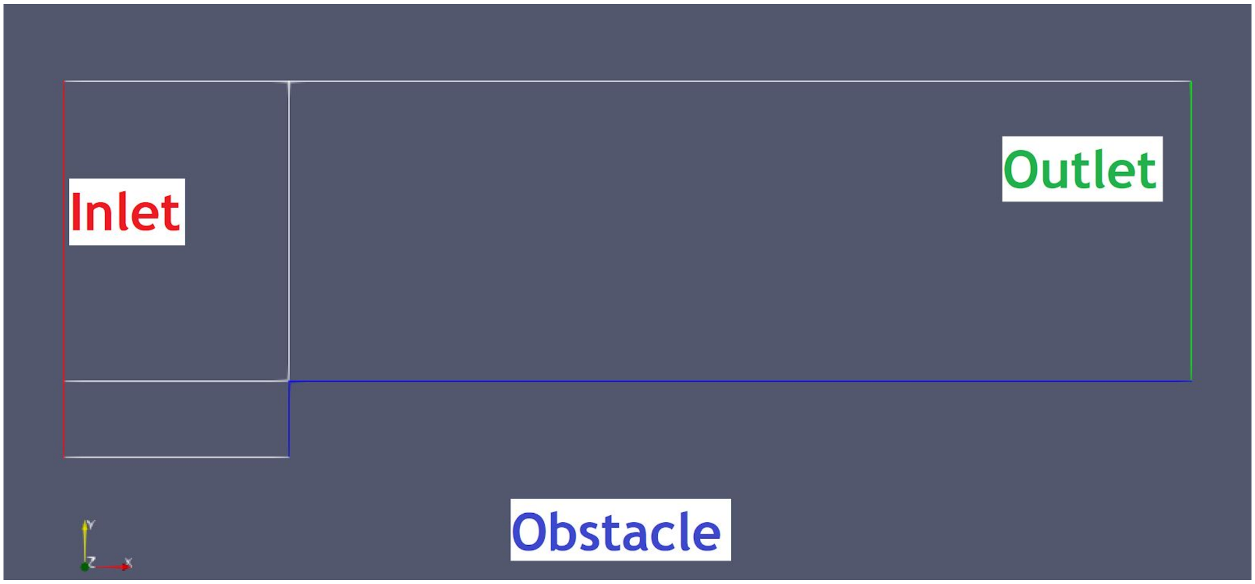


Figure 3. Schematic of boundary types.

For the following experiments, we used two meshes of different densities. The meshes will be referred to as mesh A and mesh B. The technical details of mesh A and B are tabulated in table 1 below.

Mesh	Bounding box	Number of points	Number of cells	Number of faces	Grid spacing
A	(0 0 -0.05) (3 1 0.05)	51592	25410	102025	uniform
B	(0 0 -0.05) (3 1 0.05)	95552	47250	189525	uniform

Table 1. Details of meshes A and B.

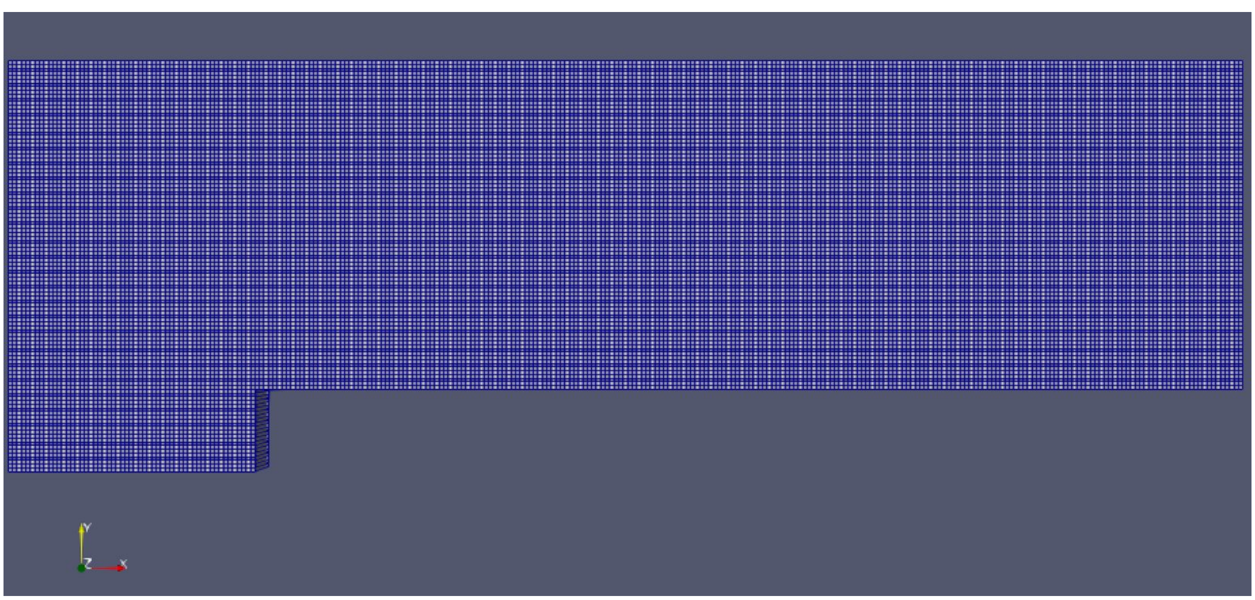
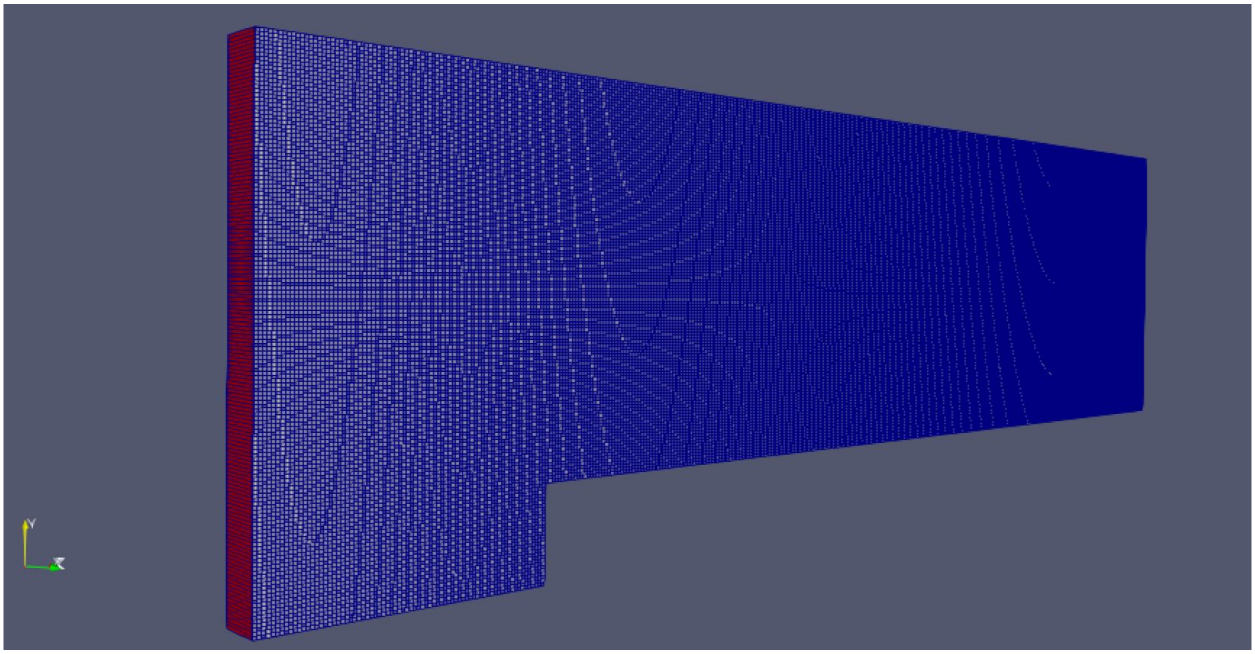


Figure 4. Image of Mesh A (51,592 points).



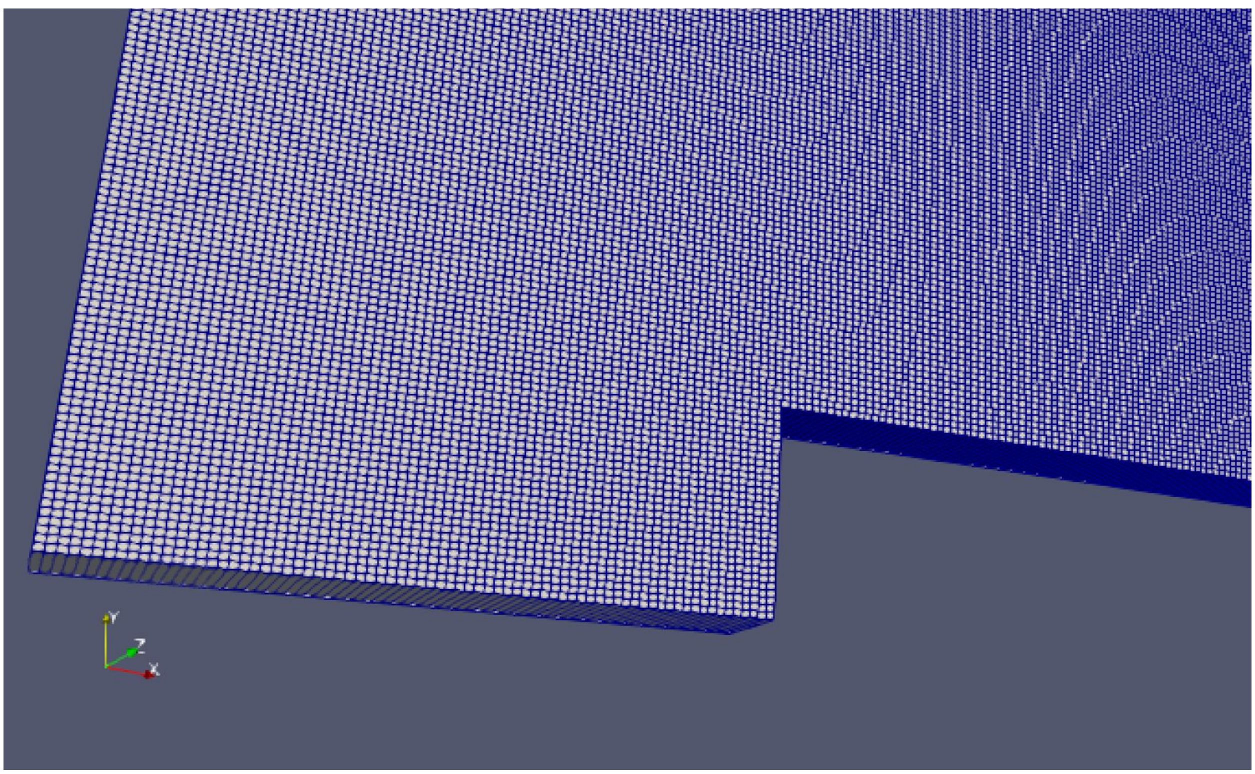


Figure 5. Images of Mesh B (95,552 points).

Convergence with rhoCentralFoam

We ran the rhoCentralFoam simulations with the two mesh densities described in table 1. We used $\Delta t = 0.002$ for all trials.

<u>Results with Mesh A (51,592 Points)</u>

Figure 6. Plot of the magnitude of velocity at t=4s on a 2D section of mesh A.

Figure 7. Plot of velocity in the x-direction at t=4s on a 2D section of mesh A.

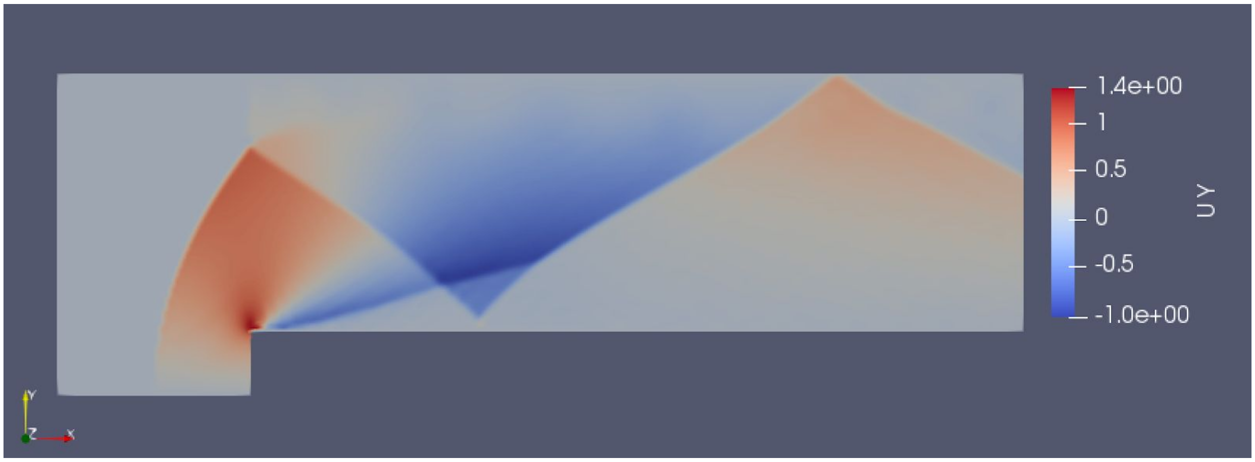


Figure 8. Plot of velocity in the y-direction at t=4s on a 2D section of mesh A.

Figure 9. Plot of pressure at t=4s on a 2D section of mesh A.

Figure 10. Plot of temperature at t=4s on a 2D section of mesh A.

Figure 11. Plot of rho at t=4s on a 2D section of mesh A.

Figure 12. Plot of a at t=4s on a 2D section of mesh A. a was calculated as $a = \sqrt{1.4 * \frac{p}{\rho}}$.

Figure 13. Plot of Mach number at t=4s on a 2D section of mesh A. Mach number was calculated as $M = \frac{|u|}{a}$ where |*u*| is the magnitude of velocity and *a* is calculated as $a = \sqrt{1.4 * \frac{p}{\rho}}$.

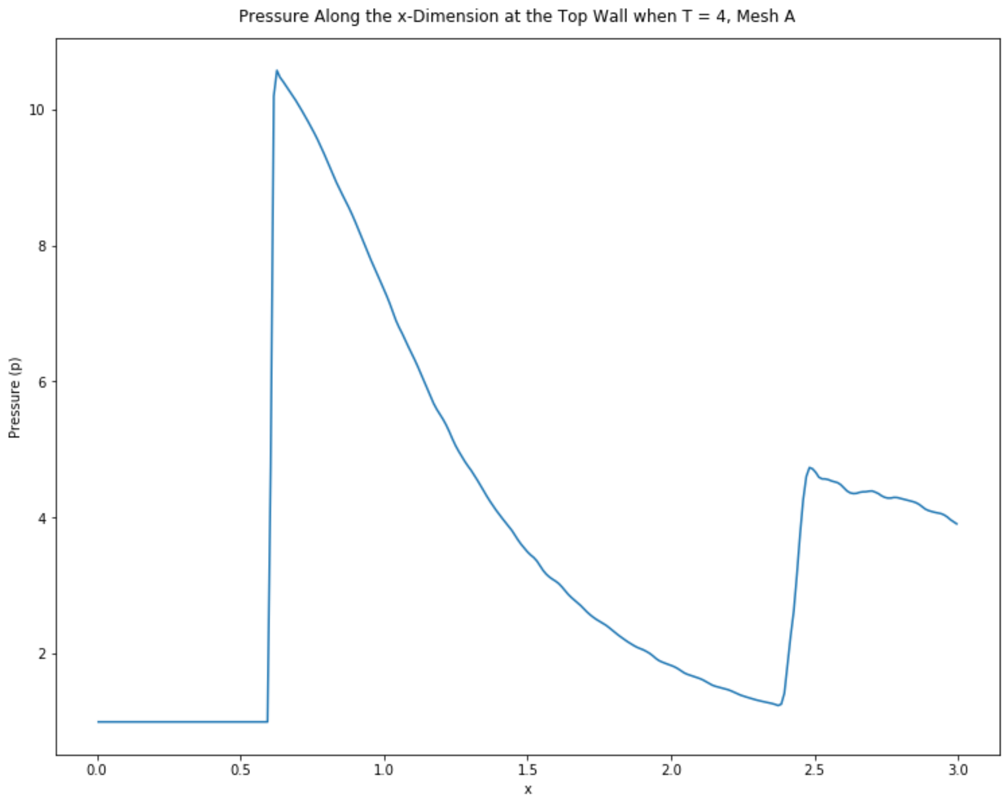


Figure 14. Plot of pressure on the top wall along the x axis of mesh A at t=4.

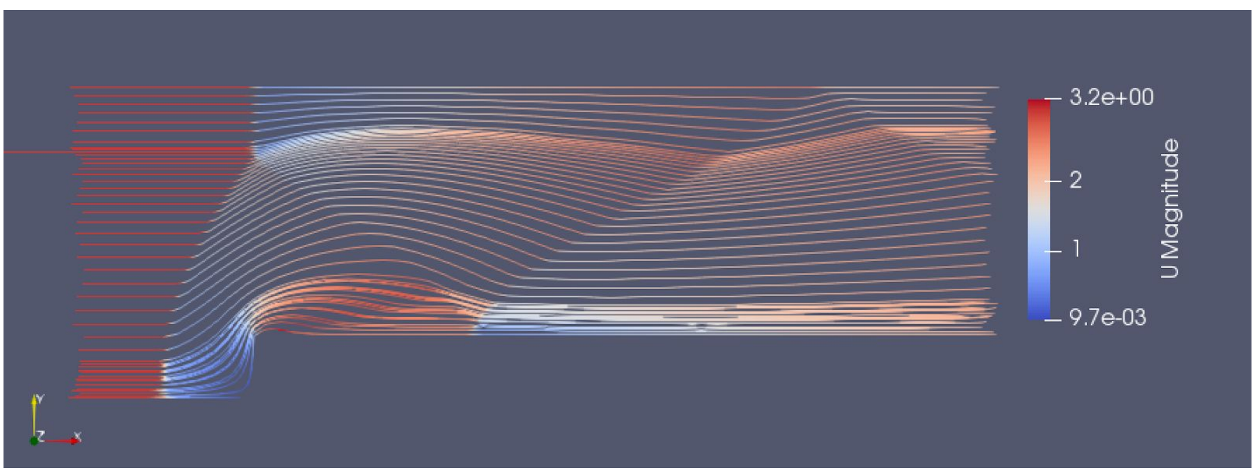


Figure 15. Plot of streamlines (color coded by velocity magnitude) at t=4s on a 2D section of mesh A.

Mesh B (95,552 Points)

Figure 16. Plot of the magnitude of velocity at t=4s on a 2D section of mesh B.

Figure 17. Plot of the velocity in the x-direction at t=4s on a 2D section of mesh B.

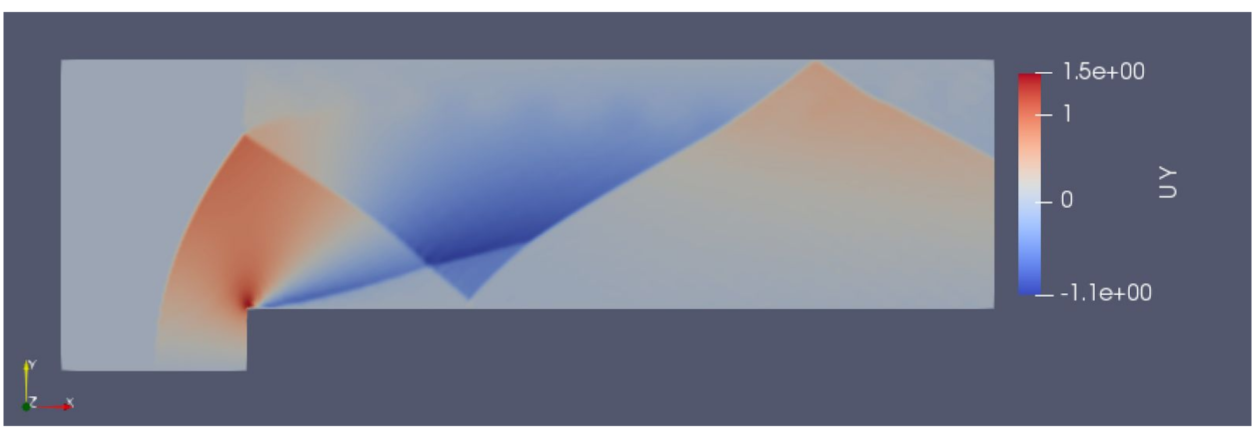


Figure 18. Plot of the velocity in the y-direction at t=4s on a 2D section of mesh B.

Figure 19. Plot of pressure at t=4s on a 2D section of mesh B.

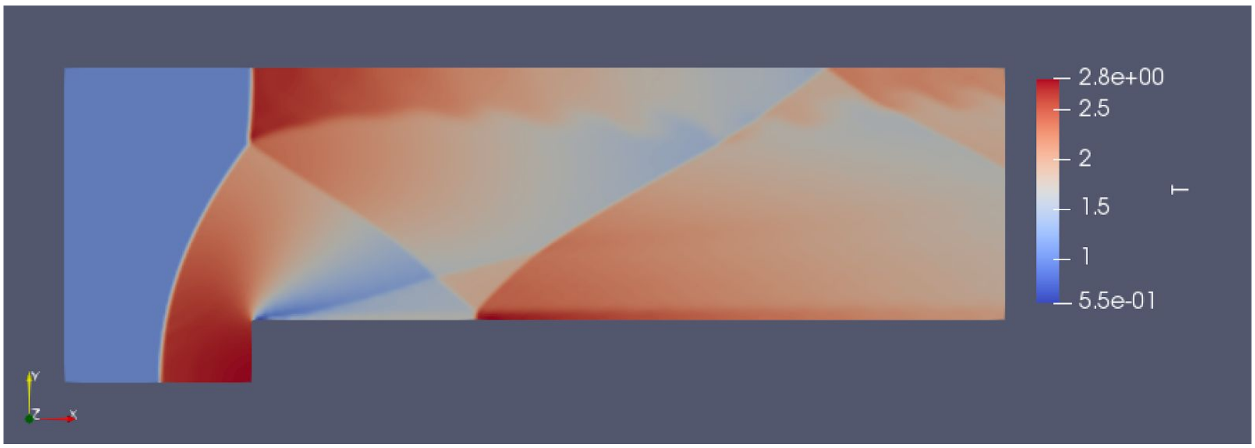


Figure 20. Plot of temperature at t=4s on a 2D section of mesh B.

Figure 21. Plot of rho at t=4s on a 2D section of mesh B.

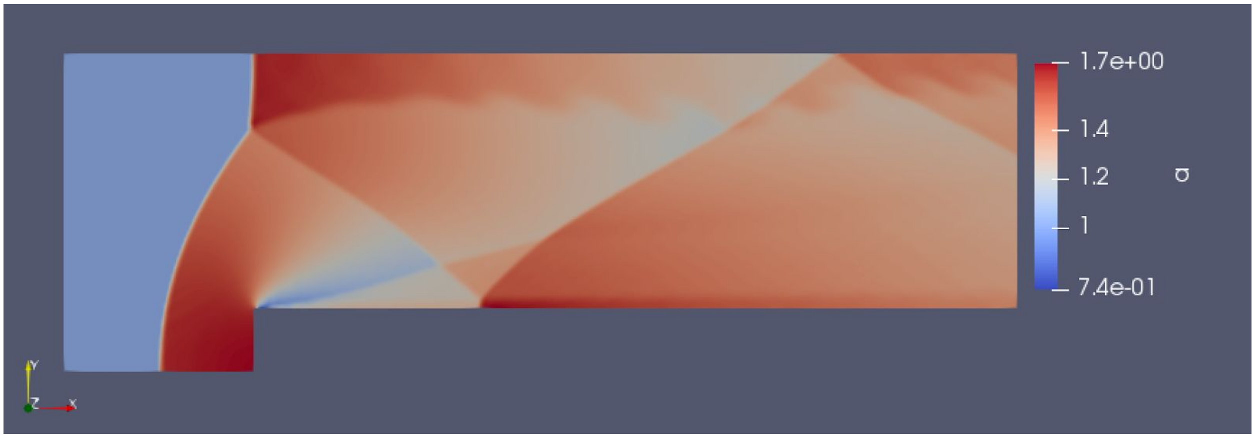


Figure 22. Plot of a at t=4s on a 2D section of mesh B. a was calculated as $a = \sqrt{1.4 * \frac{p}{\rho}}$.

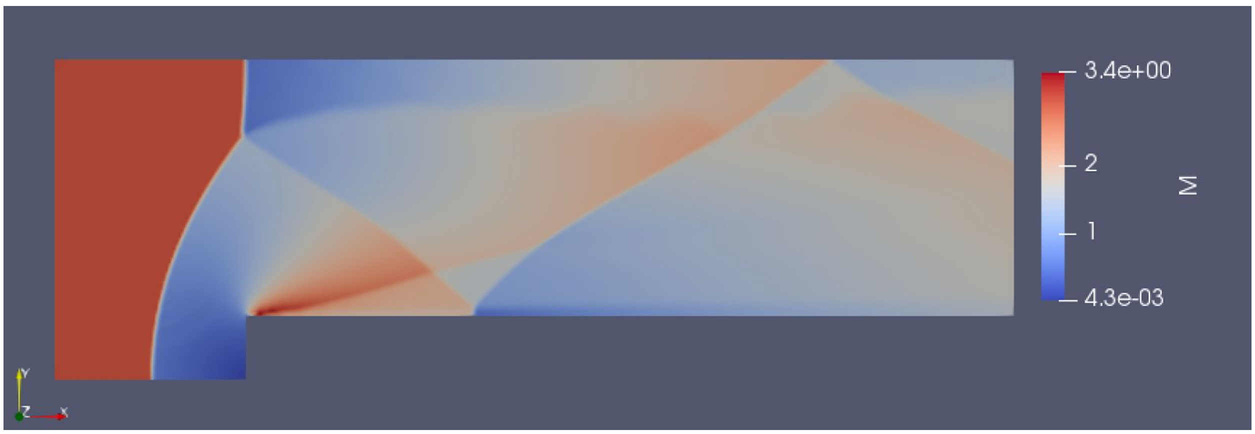


Figure 23. Plot of Mach number at t=4s on a 2D section of mesh B. Mach number was calculated as $M = \frac{|u|}{a}$ where |*u*| is the magnitude of velocity and *a* is calculated as $a = \sqrt{1.4 * \frac{p}{\rho}}$.

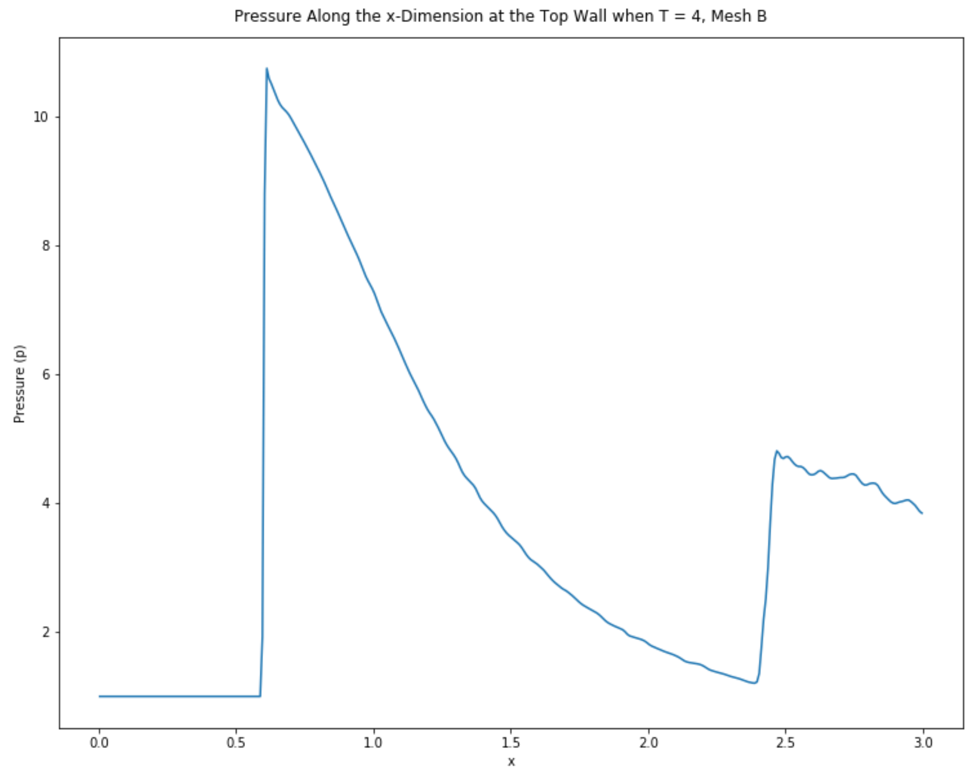


Figure 24. Plot of pressure on the top wall along the x axis of mesh B at t=4.

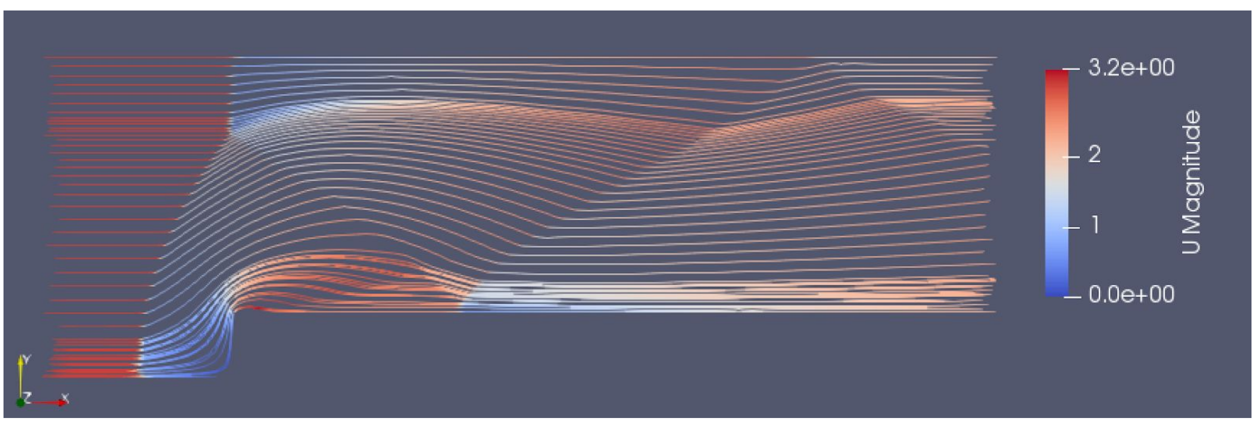


Figure 25. Plot of streamlines (color coded by velocity magnitude) at t=4s on a 2D section of mesh B.

Compared to our simulation results, we see that the pressure approximations from the normal shock tables quite accurately predicted the pressure profile of the shock. With an observed downstream pressure approximately 10 times greater than the upstream pressure and a constant $\frac{dP}{dx}$ across the shock, the simulation was able to capture the complex physical nature of a shock with considerable precision.

Extra Credit: Converge the Solution with a Faster Solver

We performed the same simulations with sonicFoam, an OpenFOAM solver. RhoCentralFoam and sonicFoam use different numerical integration schemes. Specifically, sonicFoam uses pressure and velocity as dependent variables and uses a PISO (Pressure-Implicit with Splitting of Operators) method. This is different from rhoCentralFoam. rhoCentralFoam uses a central-upwind scheme.

To gain an understanding of the difference in clock times between the two methods, we ran each solver on mesh A 3 times. The averaged results are tabulated below.

Average Wall Clock Times For Mesh A	
Solver	Average Wall Clock Time (s)
rhoCentralFoam	478s
sonicFoam	186s

Clearly, the sonicFoam solver executed in less time. We visualized the magnitude of the velocity for each solver and analyzed the accuracy of each solver below.

<u>rhoCentralFoam Solver</u>

Figure 26. Plot of the magnitude of velocity at t=2s on a 2D section of mesh A. Solved with the rhoCentralFoam solver.

Figure 27. Plot of the magnitude of velocity at t=4s on a 2D section of mesh A. Solved with the rhoCentralFoam solver.

sonicFoam Solver

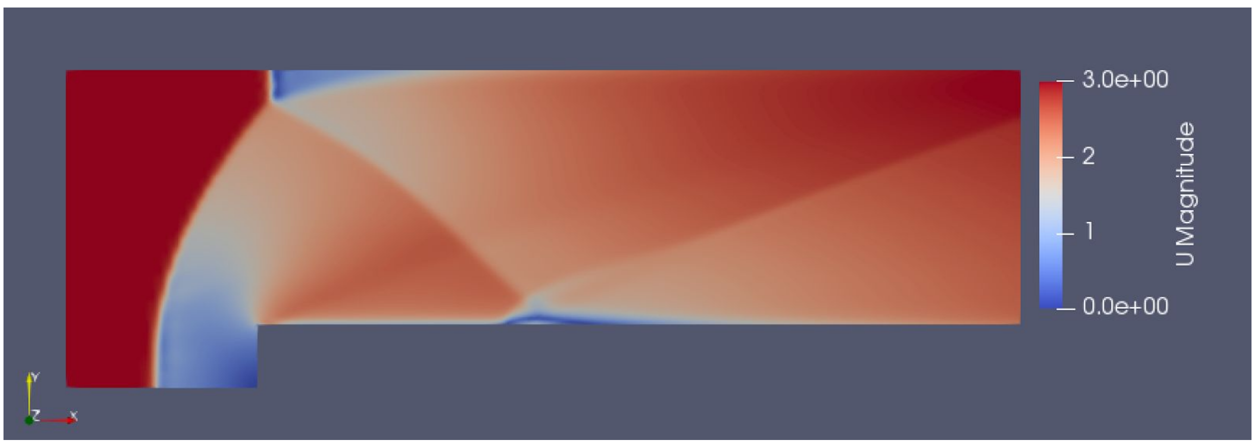


Figure 28. Plot of the magnitude of velocity at t=2s on a 2D section of mesh A. Solved with the sonicFoam solver.

Figure 29. Plot of the magnitude of velocity at t=4s on a 2D section of mesh A. Solved with the sonicFoam solver.

Clearly, the flow developed differently with the sonicFoam solver. To understand the causes of the differences we researched the differences between the two solvers. After research, we concluded that rhoCentralFoam likely provides a more accurate solution than sonicFoam for our supersonic flow simulations.

In compressible flows, properties must be transported by the flow velocity as well as the propagation of waves. For that reason, the central-upwind scheme used by rhoCentralFoam has been shown to be one of the schemes for various types of supersonic flows (Bondarev).

SonicFoam was developed using the PISO method. The PISO method was originally designed to model incompressible flows (Gutierrez). After it was created the PISO method was adapted to work with compressible flows. However, the PISO method is still a density based solver, and works best with flows near the incompressible limit of $M < 0.3$. Since our flow is compressible with $M = 3$, sonicFoam likely is not very accurate.

Sources

Bondarev, Alexander E & Kuvshinnikov, Artem E. (2018). *Analysis of the Accuracy of OpenFOAM Solvers for the Problem of Supersonic Flow Around a Cone*.

Gutierrez Marcantoni, Luis & Tamagno, José & Elaskar, Sergio. (2012). *High Speed Flow Simulation Using openFOAM*.