THE UNIVERSITY OF BRITISH COLUMBIA OKANAGAN CAMPUS

FACULTY OF APPLIED SCIENCE, SCHOOL OF ENGINEERING

ENGR 491

COMPUTATIONAL FLUID DYNAMICS

SE 6: SUPERSONIC FLOW OVER A WEDGE

SUBMITTED BY:

NICHOLAS SABRY 38029161

Problem Description: A description of the problem, computational domain, boundary conditions, and meshing strategy.

Answer: Description of the problem

A uniform supersonic stream interacts with a wedge with $\theta = 15$ degrees as shown in Figure 1. The stream conditions are:

Mach Number $M_1 = 3$ Static Pressure $p_1 = 1$ Static Temperature $T_1 = 1\ k$

Calculate the theoretical values for $M_2, p_2,$ and T_2 after the shock as well as the β angle created by the shock and compare them to the simulated values obtained for $M_2, p_2,$ and T_2 after the shock as well as β angle created by the shock to validate the simulation results.

Answer: Computational domain

The us of rhoCentralFoam is employed for this simulation since the flow over the 15-degree wedge at Mach 3 forms a shock. Because a shock is formed a solver that can deal with discontinues is required.

The computational domain for flow over the wedge is defined by the 5 points in Figure 1.

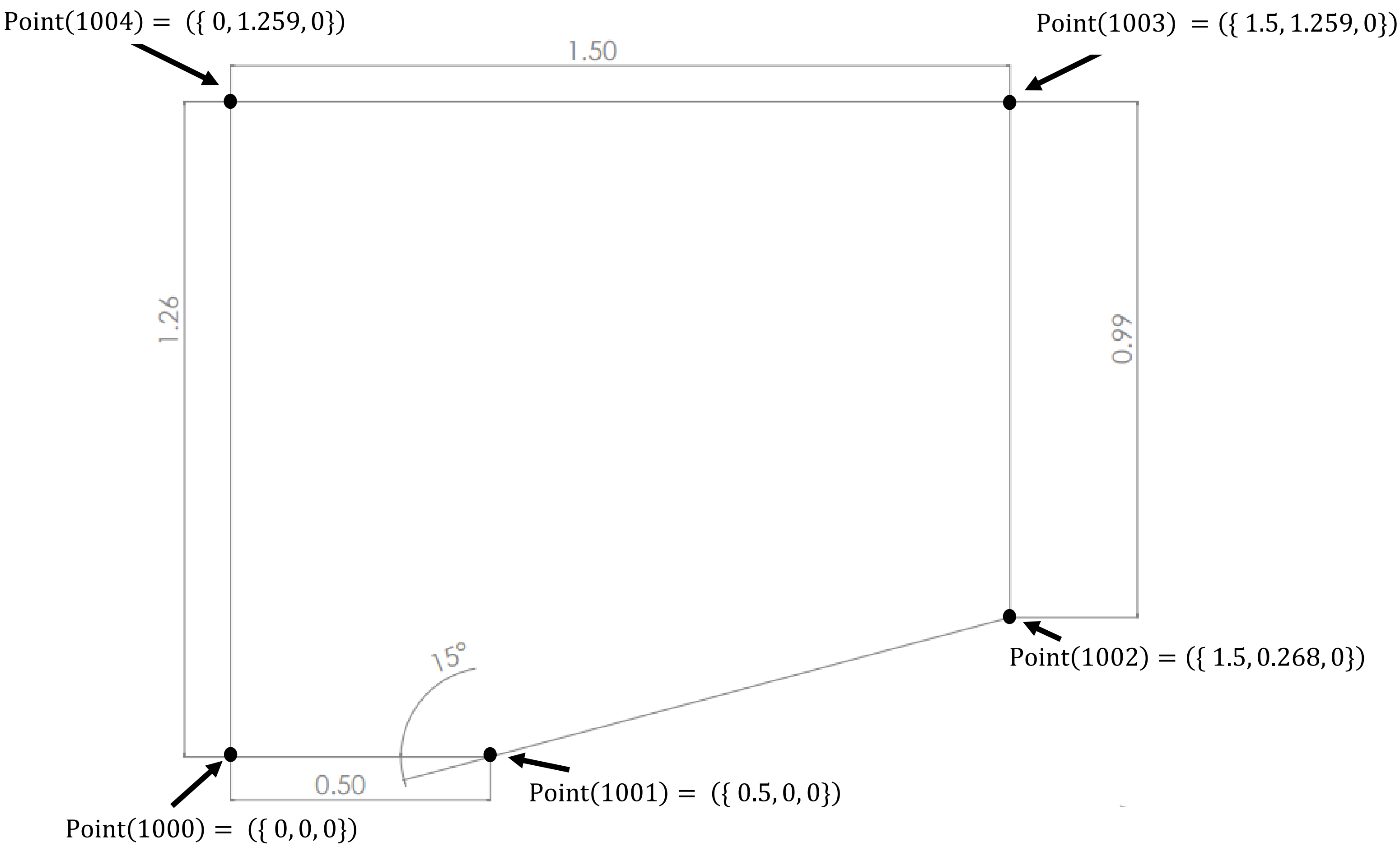


Figure 1:

Answer: Boundary Conditions

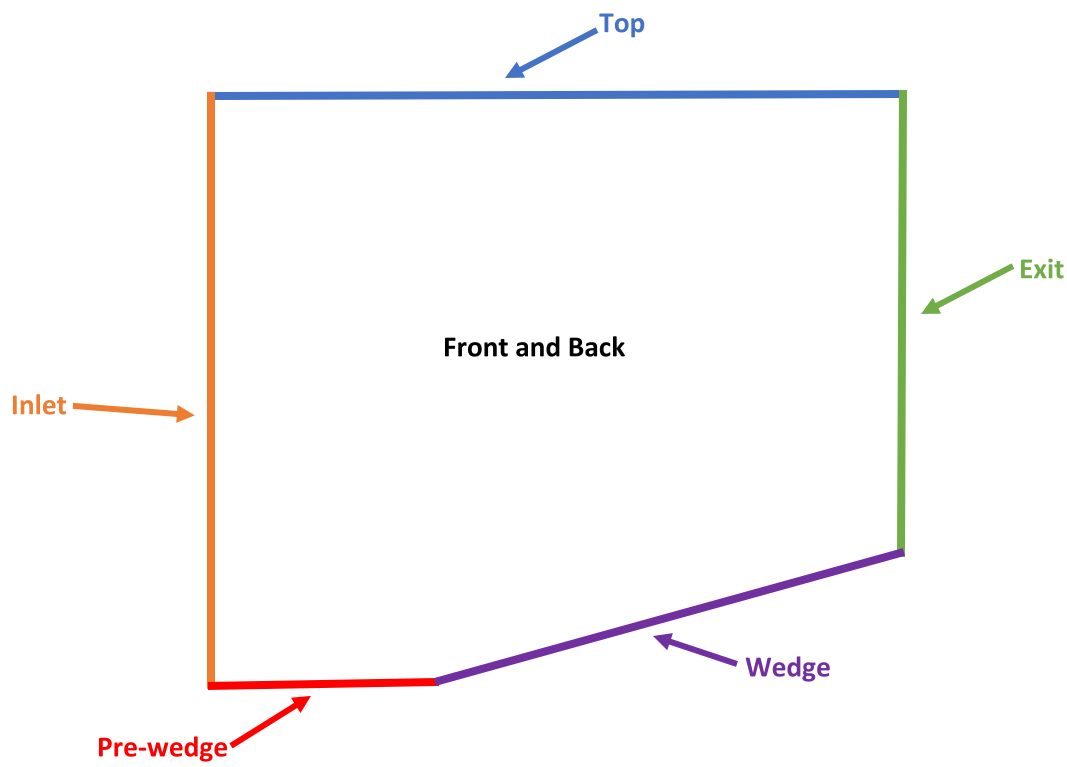


Figure 2:

<u>Pressure Boundary Conditions:</u>

Inlet: Boundary Field Type: fixed value

Exit: Boundary Field Type: zero gradient

Wedge: Boundary Field Type: zero gradient

Pre-wedge: Boundary Field Type: symmetry plane

Top: Boundary Field Type: symmetry plane

Front: Boundary Field Type: empty

Back: Boundary Field Type: empty

Temperature Boundary Conditions:

Inlet: Boundary Field Type: fixed value

Exit: Boundary Field Type: inlet outlet

Wedge: Boundary Field Type: zero gradient

Pre-wedge: Boundary Field Type: symmetry plane

Top: Boundary Field Type: symmetry plane

Front: Boundary Field Type: empty

Back: Boundary Field Type: empty

Velocity Boundary Conditions:

Inlet: Boundary Field Type: fixed value

Exit: Boundary Field Type: inlet outlet

Wedge: Boundary Field Type: fixed value

Pre-wedge: Boundary Field Type: symmetry plane

Top: Boundary Field Type: symmetry plane

Front: Boundary Field Type: empty

Back: Boundary Field Type: empty

Answer: Meshing Strategy

The 5 points are inputted into .geo file and the following text was created to define the wedge in Gmsh

```
// SupersonicFlowOveraWedge
ss_lc = 0.005;
edge_lc = 0.5;
 Point(1000) = { 0.00000000, 0.00000000, 0.00000000, ss_lc};
 Point(1001) = { 0.50000000, 0.00000000, 0.00000000, ss_lc};
 Point(1002) = { 1.50000000, 0.26800000, 0.00000000, ss_lc };
 Point(1003) = { 1.50000000, 1.25900000, 0.00000000, ss_lc };
 Point(1004) = { 0.00000000, 1.25900000, 0.00000000, edge_lc };
Line(1) = {1000,1001};
Line(2) = {1001,1002};
Line(3) = {1002,1003};
Line(4) = {1003,1004};
Line(5) = {1004,1000};
Line Loop (1) = {1,2,3,4,5};
Plane Surface(1) = {1};
Extrude {0, 0, 1} {
  Surface{1};
  Layers{1};
  Recombine;
}
Physical Surface("inlet") = {31};
Physical Surface("exit") = {23};
Physical Surface("top") = {27};
Physical Surface("front") = {1};
Physical Surface("back") = {32};
Physical Surface("wedge") = {19};
Physical Surface("prewedge") = {15};
Physical Volume("internal") = {1};
```

The points were defined Point(1000) to Point(1001) and lines were defined between all the points using Line(1) to Line(5). Then the lines were connected using Line Loop and a surface was defined from that. Since Open Foam only accepts 3D mesh's the surface was extruded out by 1 unit. Then each physical surface was defined and named. The corresponding numbers to each physical surface must be the same as the values shown in gmsh.

Using the file sequence:

Tools (Top Right) → Options (First Drop Down Item) → Geometry (Second Drop Down Item) →

Visibility (Second Item Left) → Surface Labels (Tick The Box)

We get Figure 3:

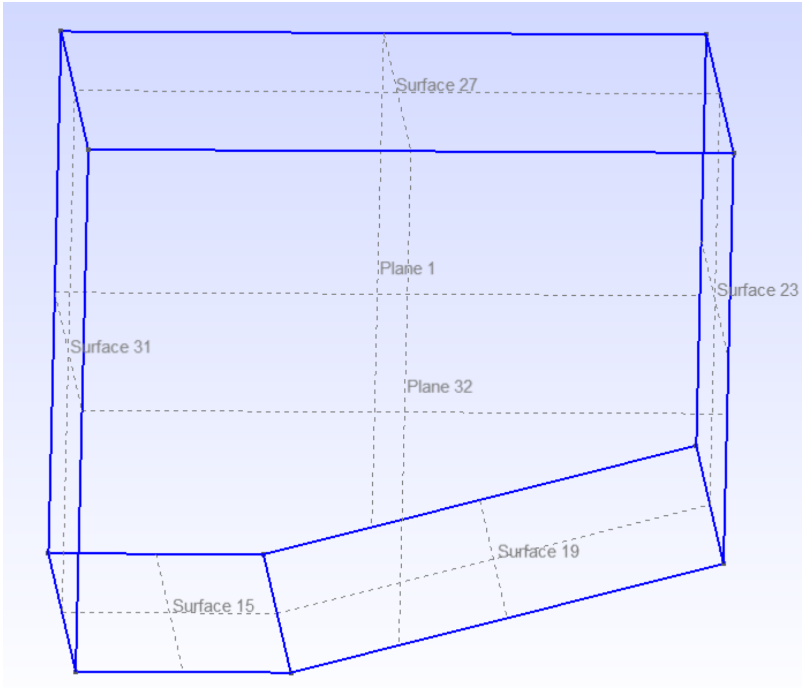


Figure 3:

Making sure the physical surfaces match the numbers on the Figure 3 is required to run the simulation on OpenFoam correctly.

Next generation of the mesh is required, Using the file sequence:

Modules (Top Right, hit Plus) → Mesh (Second Drop Down Item) → 3D (3rd Drop Down Item) →

Will generate a 3D mesh that can then be saved into a specific format that openFoam will accept, Using the file sequence:

File (Top Right) → Export (Select "Mesh – Gmsh MSH (*.msh)) → Save →

MSH Options (Click Fomat and select Version 2 ASCII) → OK

Then to convert it into a mesh file for openFoam find the directory the above .msh file was saved in and run mshToFoam “Same name of the file”.msh

The resulting mesh from these steps is shown in Figure 4.

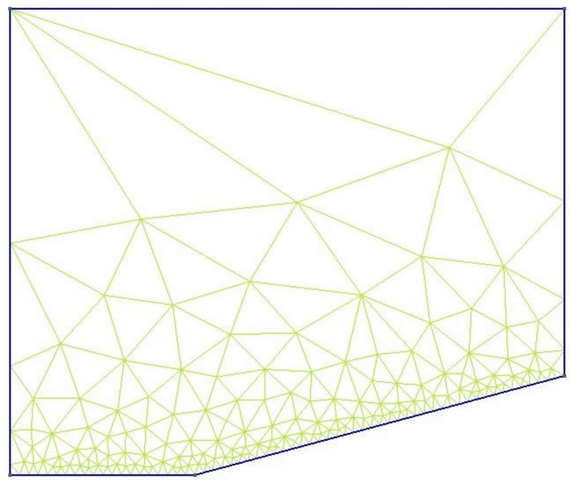


Figure 4:

This mesh is very course and needs to be refined by making all the cells smaller and by adjusting the .geo file the Exit can have finer cells compared to the far field where fine cells are not required.

Just for reference this is what the flow would have done if the current mesh was used:

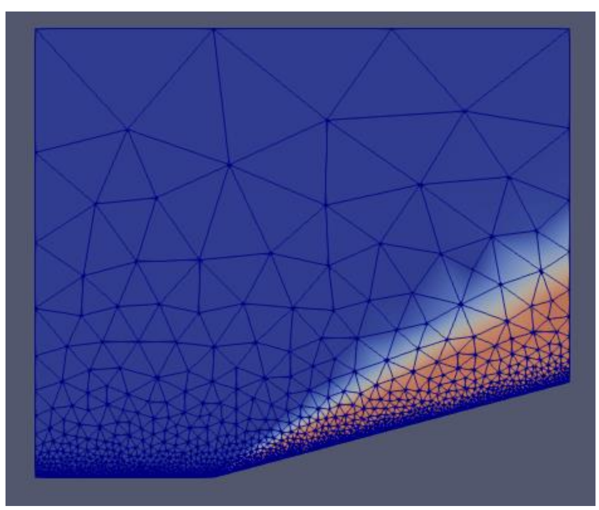


Figure 5:

As shown in Figure 5 the flow around the shock is very hard to resolve and takes the shape of the cells around that area which would give incorrect values for the Mach number:

Refining the correct highlighted numbers in the .geo file as to not affect the far field cells:

```
// SupersonicFlowOveraWedge
ss_lc = 0.005;
edge_lc = 0.5;
 Point(1000) = { 0.00000000, 0.00000000, 0.00000000, ss_lc};
 Point(1001) = { 0.50000000, 0.00000000, 0.00000000, ss_lc};
 Point(1002) = { 1.50000000, 0.26800000, 0.00000000, ss_lc };
 Point(1003) = { 1.50000000, 1.25900000, 0.00000000, ss_lc };
 Point(1004) = { 0.00000000, 1.25900000, 0.00000000, edge_lc };
```

Along with a global mesh factor adjustment using the following file sequence:

Double Left Click (On Background) → Global mesh size factor (Select number) → Save

Resulting in this mesh:

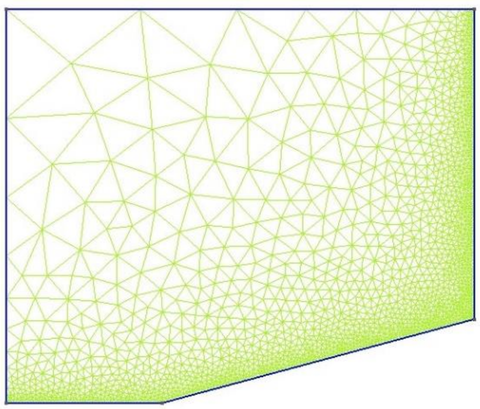


Figure 6:

Just for reference this is what the flow looks like with the mesh shown in Figure 6:

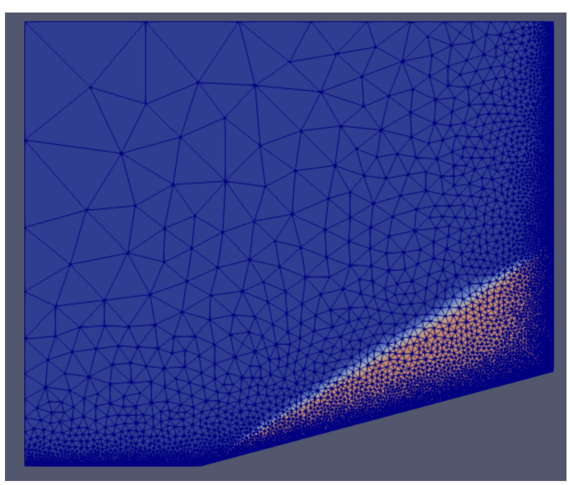
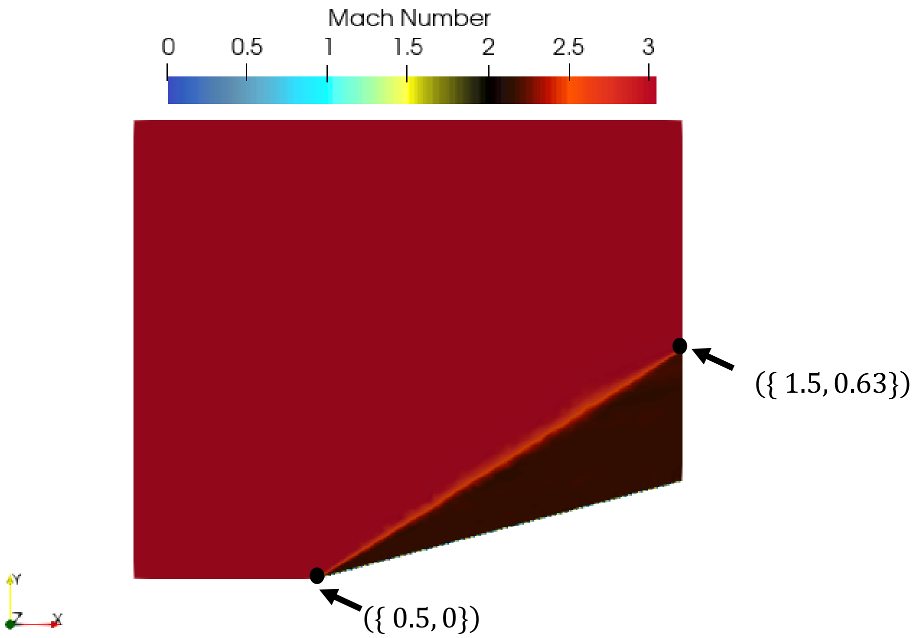


Figure 7:

As you can see with the unrefined grid in Figure 5 the flow is very hard to resolve. The first attempt at running the refined mesh, the courant number increased rapidly and stopped the simulation. An attempt to solve this issue was to decrease the time step. Decreasing the timestep allowed the simulation to run for a little longer but the simulation still stopped. Since decreasing the time step wasn't working another option is to use a less refined grid. With a less refined grid and small time step the courant number stayed below its threshold and the simulation finished the result is shown in Figure 7.

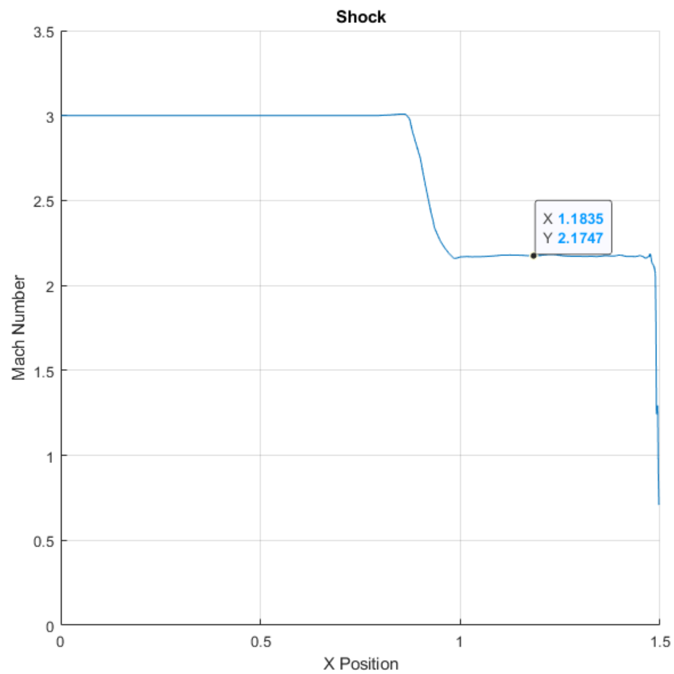
Problem 1: Presentation of the results. Present the results in a manner that describes the problem and illustrates interesting features of the flow.

Answer:

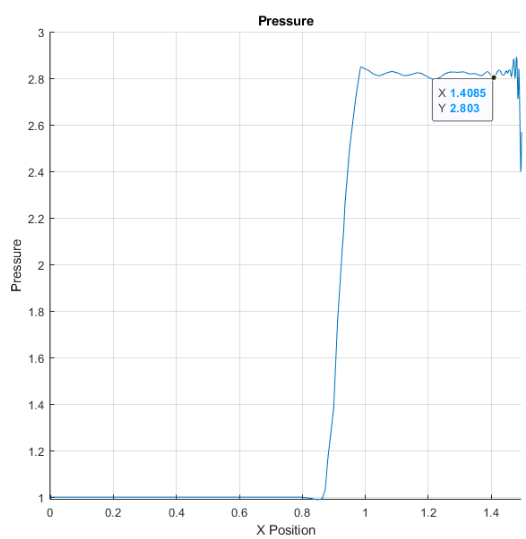


$$\sin^{-1}\left(\frac{0.63}{\sqrt{0.63^2+1^2}}\right) = 32.2°$$

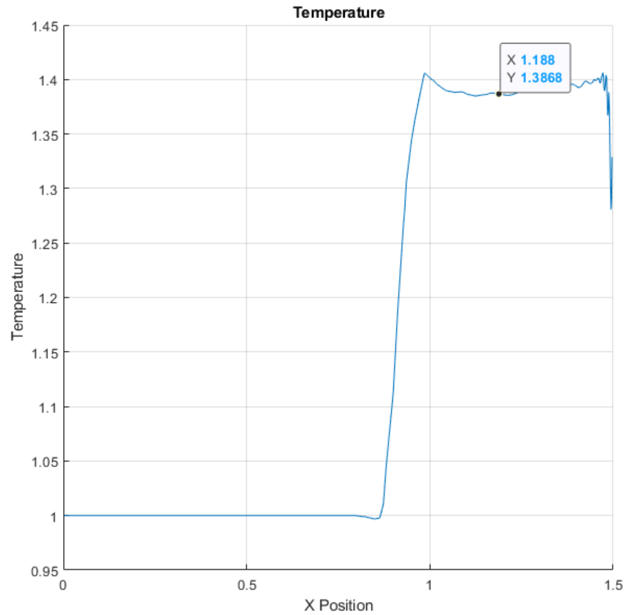
$$\boldsymbol{\beta = 32.2°}$$



$M_2 = 2.1747$



$p_2 = 2.803$

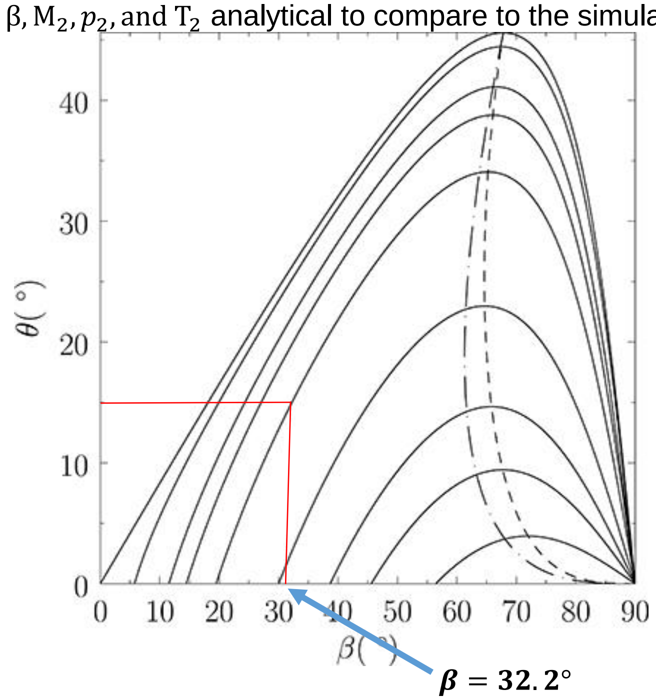

Temperature
X 1.188
Y 1.3868
Temperature
1.45
1.4
1.35
1.3
1.25
1.2
1.15
1.1
1.05
1
0.95
0
0.5
1
1.5
X Position

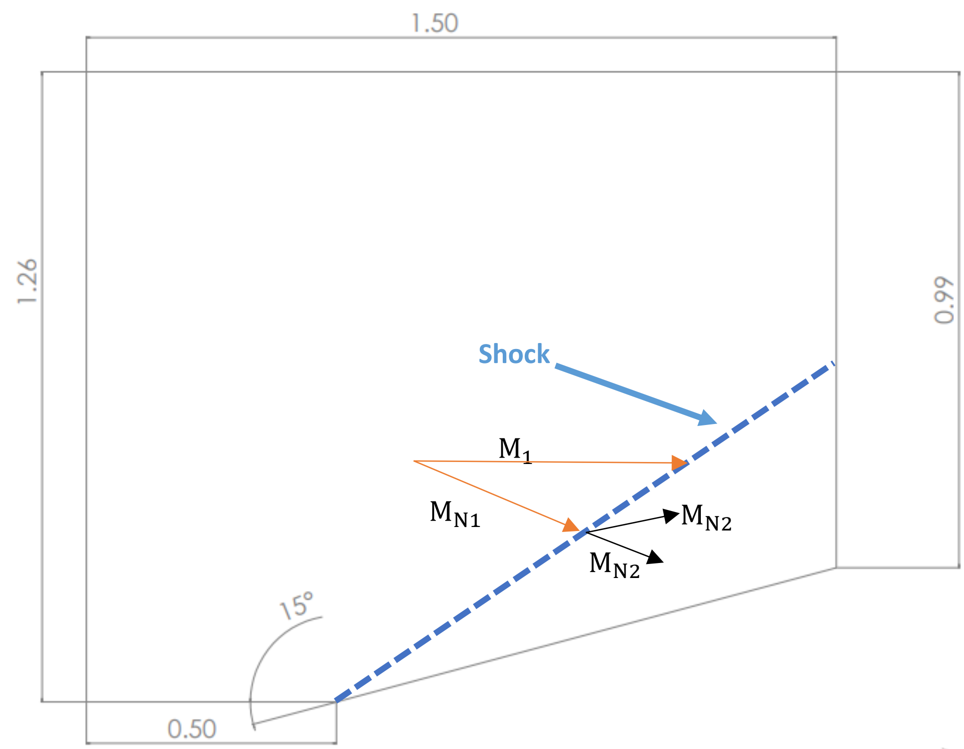
$$T_2 = 1.3868$$

Problem 2: Validation of the results against benchmark experimental, numerical, or analytical results. Discuss the accuracy of the simulation as compared to the benchmark results, and comment on what could be causing any observed inaccuracies in the solution.

Answer:

Calculating $\beta, M_2, p_2,$ and T_2 analytical to compare to the simulation results:





$$C_v = 1.78571 \quad J/kgK$$

$$R = 0.714286 \quad J/kgK$$

$$\gamma = \frac{R}{C_v} + 1 \rightarrow \gamma = \frac{0.714286}{1.78571} + 1 \rightarrow \gamma = 1.4$$

$$M_{1N} = M_1 sin(\beta) \rightarrow M_{1N} = 3 * sin(32.2) \rightarrow M_{1N} = 1.59863$$

$$M_{2N} = \sqrt{\frac{(k-1)M_{1N}^2 + 2}{2kM_{1N}^2 - k + 1}} \rightarrow M_{2N} = \sqrt{\frac{(1.4-1) * 1.59863^2 + 2}{2(1.4)(1.59863)^2 - 1.4 + 1}} \rightarrow M_{2N} = 0.66885$$

$$M_{2N} = 0.66885$$

$$M_2 = \frac{M_{2N}}{\sin(\beta - \theta)} \rightarrow M_2 = \frac{0.66885}{\sin(32.2 - 15)} \rightarrow \boldsymbol{M_2 = 2.262}$$

$$\frac{p_2}{p_1} = \frac{(k+1)M_{1N}^2}{2 + (k-1)M_{1N}^2} \rightarrow \frac{p_2}{p_1} = \frac{(1.4+1)(1.59863)^2}{2 + (1.4-1)(1.59863)^2} \rightarrow \frac{p_2}{p_1} = 2.02944 \rightarrow \boldsymbol{p_2 = 2.02944}$$

$$\frac{T_2}{T_1} = [2 + (1.4-1)(1.59863)^2] \frac{2(1.4)(1.59863)^2 - 1.4 + 1}{(1.4+1)^2(1.59863)^2} \rightarrow \frac{T_2}{T_1} = 1.38702 \rightarrow \boldsymbol{T_2 = 1.38702}$$

	$\boldsymbol{\beta}$	$\boldsymbol{M_2}$	$\boldsymbol{p_2}$	$\boldsymbol{T_2}$
Theory Value	$\mathbf{32.2}$	$\mathbf{2.262}$	$\mathbf{2.029}$	$\mathbf{1.387}$
Simulation Value	$\mathbf{32.2}$	$\mathbf{2.175}$	$\mathbf{2.803}$	$\mathbf{1.387}$
Percent Difference	**0%**	**3.85%**	**38.15%**	**0%**

The inaccuracies from the simulation especially when considering pressure arise from the fact that there is an abrupt discontinuity in the middle of the mesh. That means there is a lack of resolution across the shock. So, where the pressure might have decreased gradually over the shock it isn’t represented in the simulation. This issue could be fixed if the mesh was refined in the region of the shock and could substantially reduce the error between the simulated and theoretical value for the pressure value.