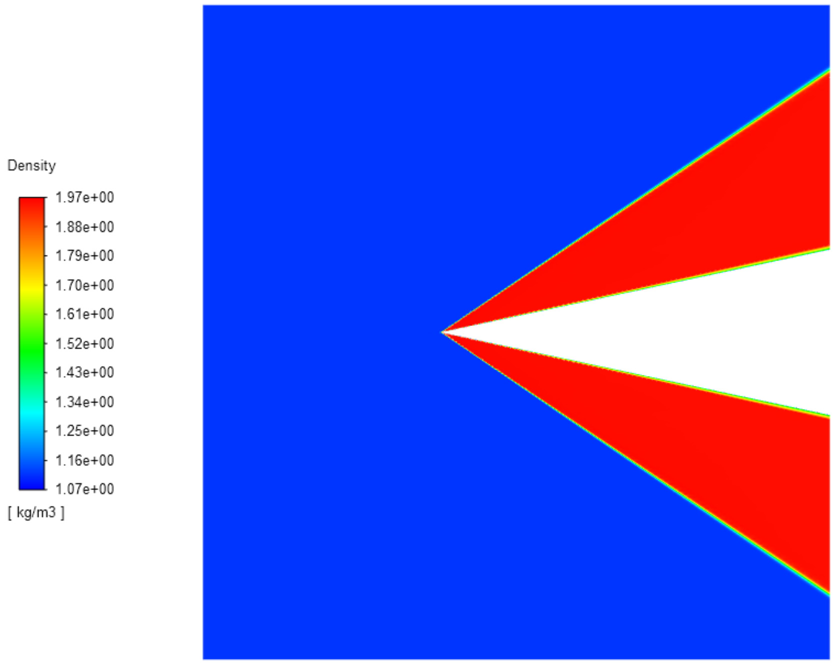
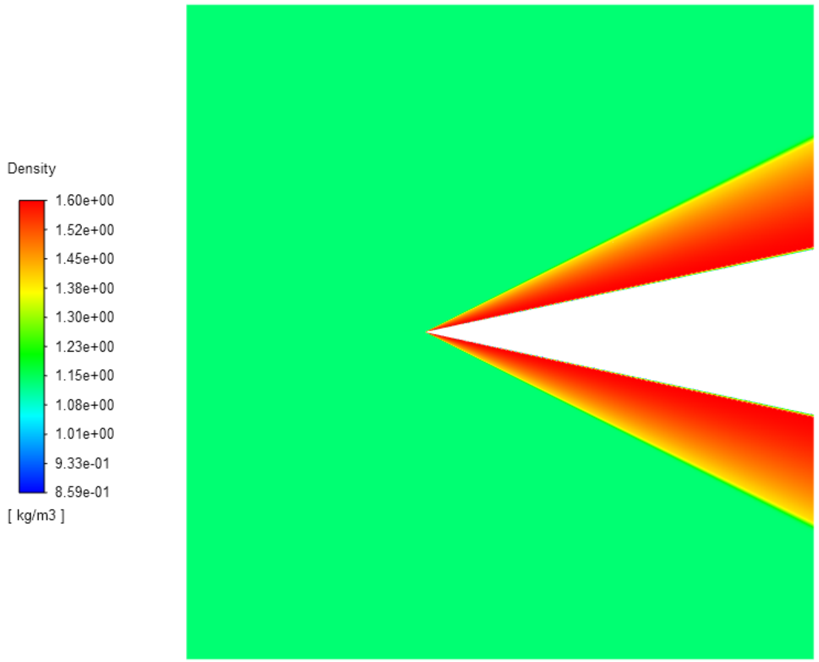
Supersonic Flow Over a Wedge and a Cone

Results

- The density contours help us visualize the shock. Different shock-wave is observed for the wedge ($\beta \approx 33.97°$) and for the cone ($\beta \approx 26.7°$). The cone generates a weaker shock than the wedge, because of the three-dimensional relieving effects. The fluid density downstream of the shock suddenly increases, as expected.



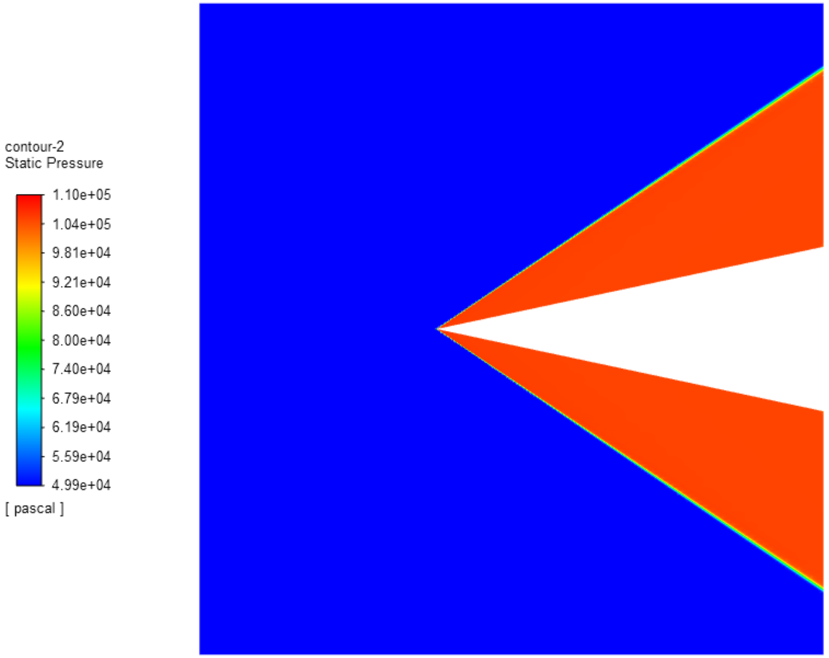
Contours of density for the wedge



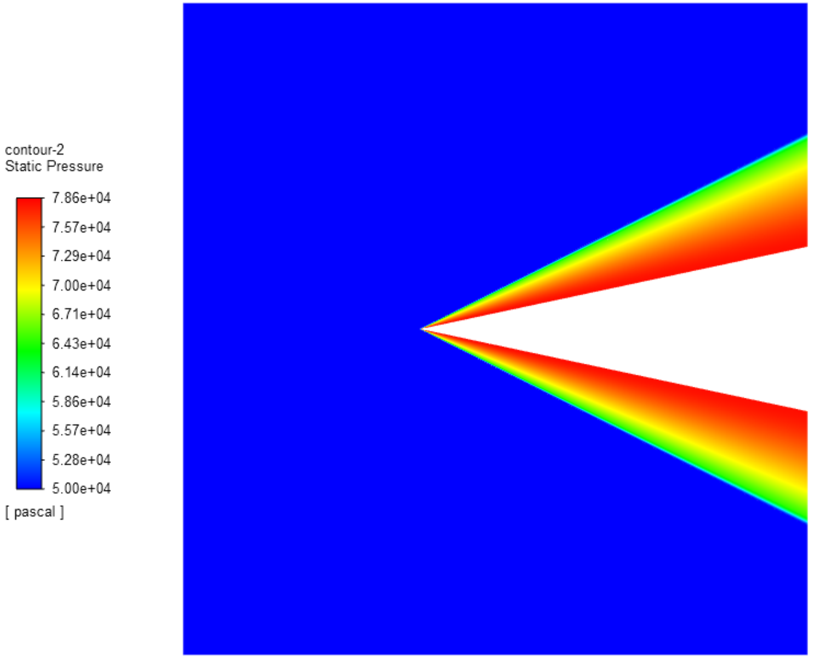
Contours of density for the cone

Results

- The static pressure contours show how the shock compresses the fluid causing a sudden rise of pressure in the wedge case. On the other hand the three-dimensional relieving effects in the cone case lead to a smaller compression across the shock, as well as a gradual increase of pressure as the flow gets closer to the cone wall.



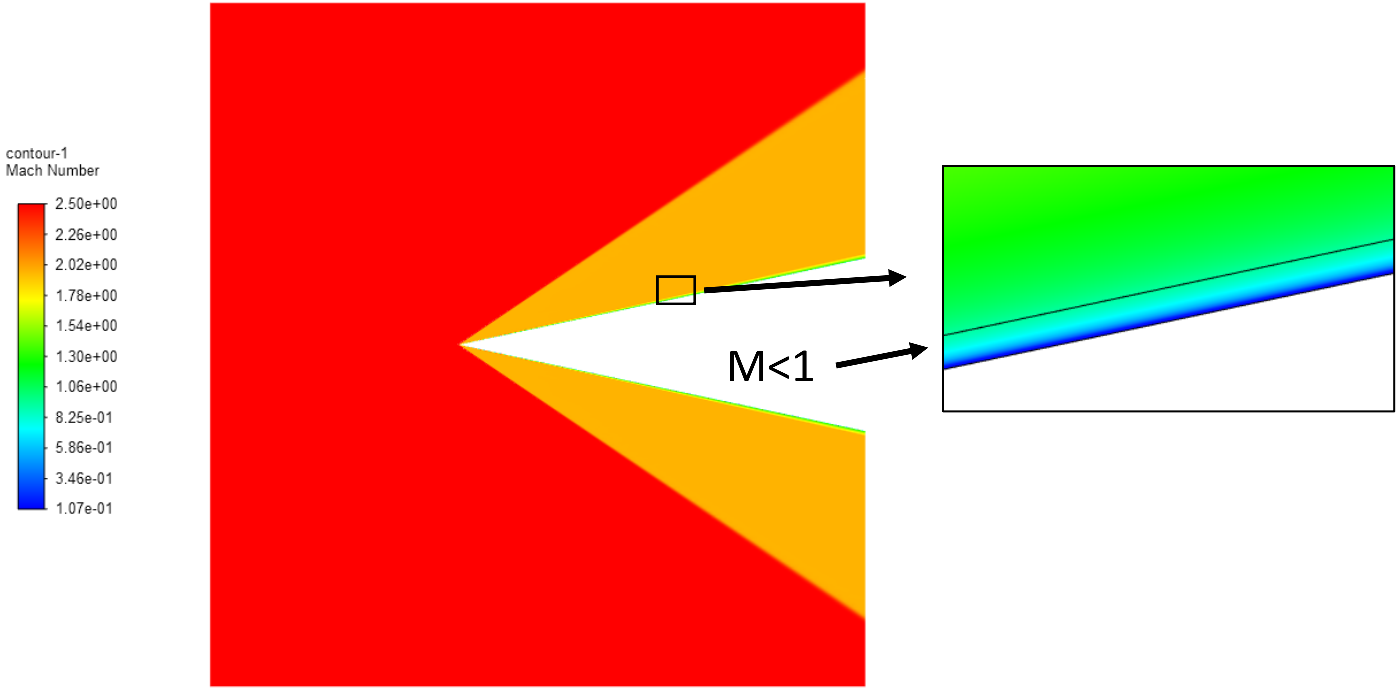
Contours of static pressure for the wedge



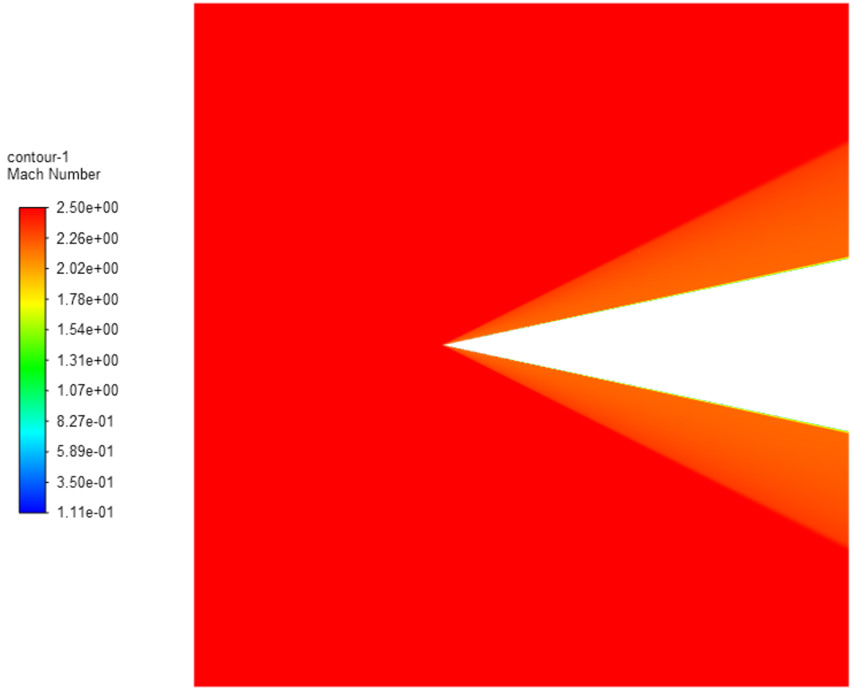
Contours of static pressure for the cone

Results

- The Mach contours highlight how the shock generated by the wedge decelerates the flow more than the cone shock. A thin boundary layers along the walls has inner subsonic layer next to the wall which transitions to supersonic in the outer part of the boundary layer.



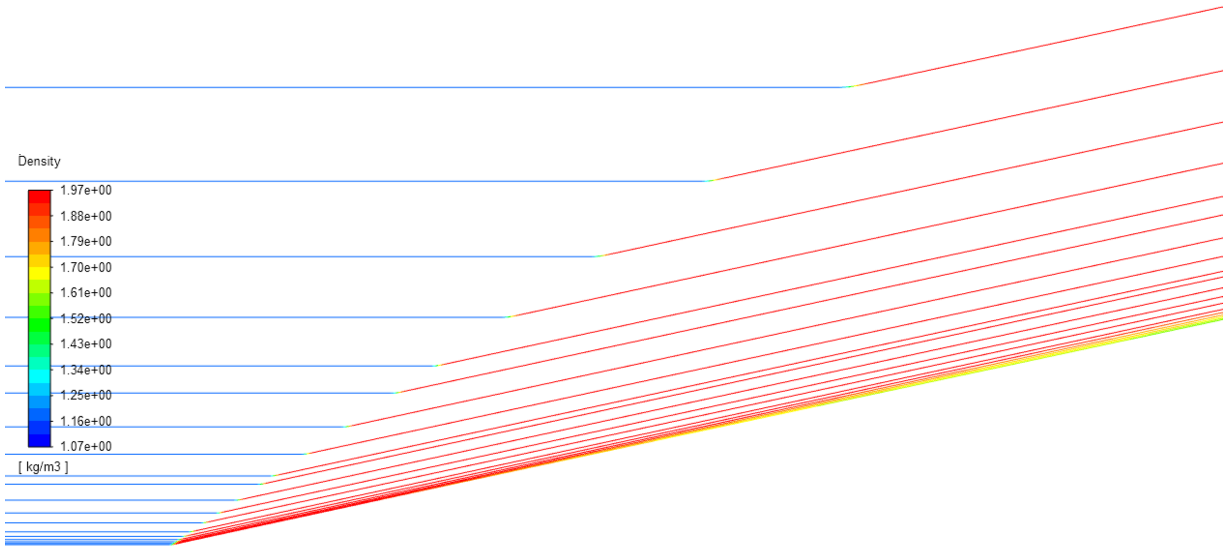
Contours of Mach number for the wedge



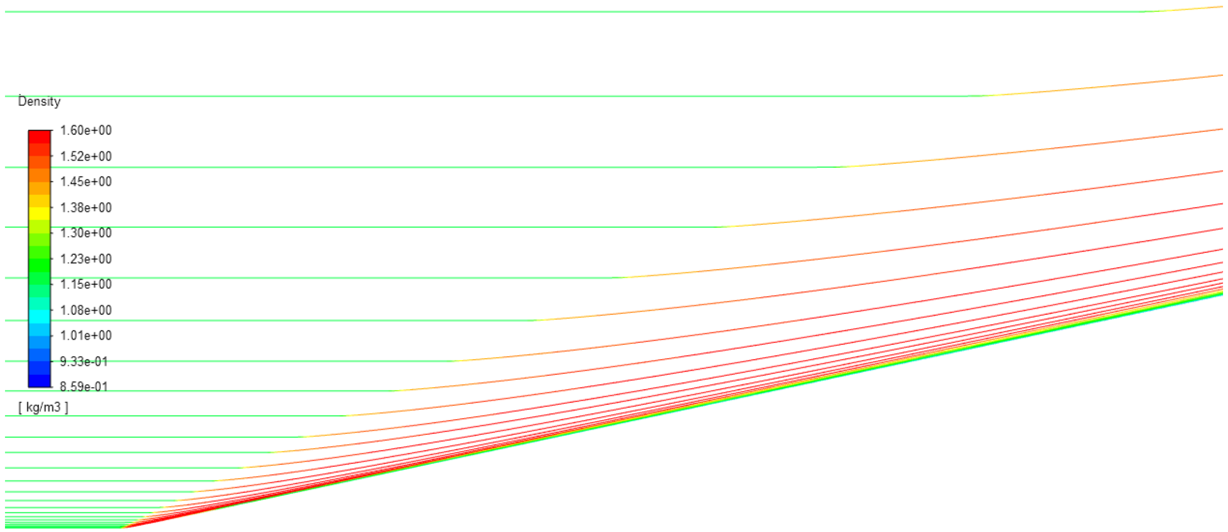
Contours of Mach number for the cone

Results

- The pathlines show how the flow suddenly changes direction past the shock generated by the wedge. The flow turns sharply by the wedge angle. On the other hand, the flow around the cone smoothly changes its direction past the shock, which is highlighted by the curvature of the pathlines.



Pathlines colored by density for the wedge



Pathlines colored by density for the cone

Results

- Comparison of the numerical results of the wedge simulation with those obtained from the oblique shock theory shows excellent agreement with the error is less than 1%:

$$M_1 = 2.5$$
$$p_1 = 50000\ Pa$$
$$\rho_1 = 1.1614\ kg/m^3$$

Oblique Shock Relations

$$M_2 = 2.002$$
$$p_2 = 104502\ Pa$$
$$\rho_2 = 1.9438\ kg/m^3$$
$$\beta = 33.80°$$

Simulations

$$M_2 = 1.995$$
$$p_2 = 105381\ Pa$$
$$\rho_2 = 1.9542\ kg/m^3$$
$$\beta = 33.97°$$

- Note that numerical data is taken from a region outside the boundary layer. The flow variables are different in the boundary layer region, due to the viscous effects that are ignored in the theory.

Results

- Comparison of the numerical results of the code simulation with those from solving the Taylor-Maccoll equation also shows excellent agreement with the error $\leq 1\%$:
-

$M_1 = 2.5$
$p_1 = 50000\ Pa$
$\rho_1 = 1.1614\ kg/m^3$

$M_2 = 2.216$
$p_2 = 77782\ Pa$
$\rho_2 = 1.5941\ kg/m^3$
$\beta = 26.4°$

Taylor-Maccoll Equation

$M_2 = 2.215$
$p_2 = 105381\ Pa$
$\rho_2 = 1.5962\ kg/m^3$
$\beta = 26.7°$

Simulations

- Note that the simulation data is taken from the region right above the boundary layer. The flow variables are different in the boundary layer region due to the viscous effects that are ignored in the theory.

Appendix

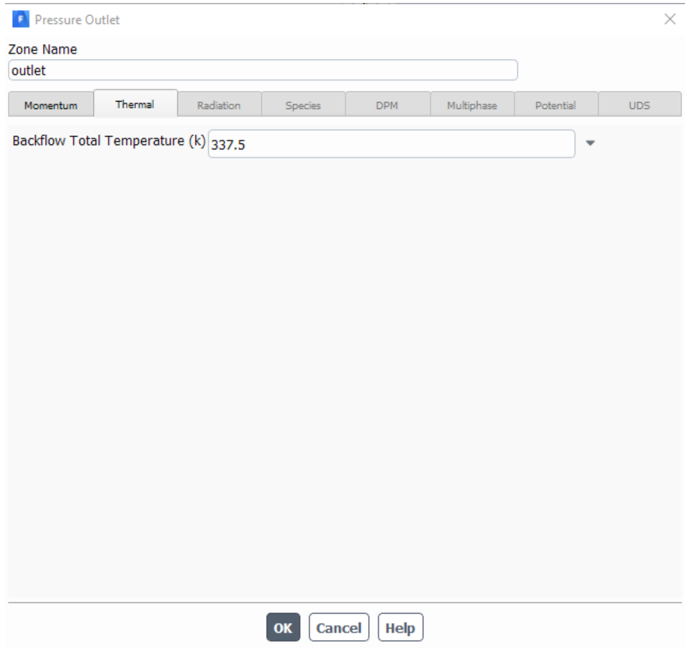
Supersonic Flow Over a Wedge and a Cone

Boundary Conditions

- The pressure far-field and the pressure outlet boundary conditions require the temperature to be set to define the conditions to solve the energy equation.
- The pressure far-field requires a static value of the temperature, while the pressure outlet requires a backflow total temperature.
- In order to estimate the total temperature we can use the isentropic relations as follows:

$$T = 150\ K \qquad M = 2.5 \qquad \gamma = 1.4$$

$$T_0 = T\left(1 + \frac{\gamma - 1}{2} M^2\right) = 337.5\ K$$



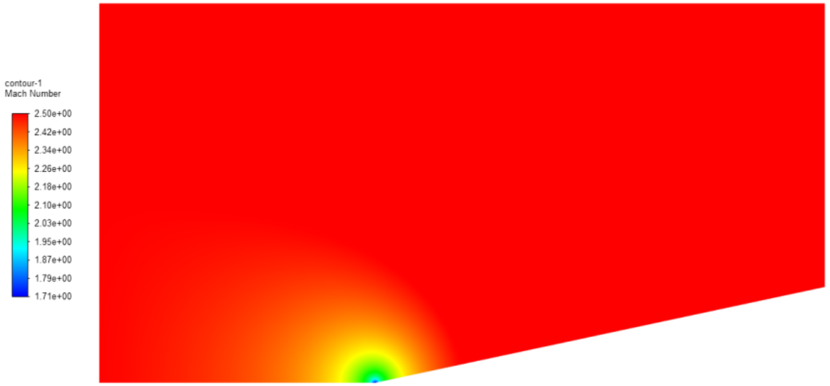
- Note that the backflow temperature will be used by the solver only if reversed flow develops at the outlet, which does not happen in this case.

FMG initialization

- The Full Multigrid (FMG) initialization is used to obtain a better initial solution in order to decrease the overall cost of the simulation.
- The FMG initialization requires an initial condition on the field to start with. Then it calculates a solution on coarser grids and interpolates it back on the main grid.
- The initialization can be performed from the Ansys Fluent Console typing the following TUI command
  - solve/initialize/fmg-initialization yes

```
Console
/solve/initialize>
compute-defaults/                list-defaults
fmg-initialization               reference-frame
hyb-initialization               repair-wall-distance
init-flow-statistics             set-defaults/
init-turb-vel-fluctuations       set-fmg-initialization
initialize-flow                  set-hyb-initialization
```

Snapshot of the Console TUI menus

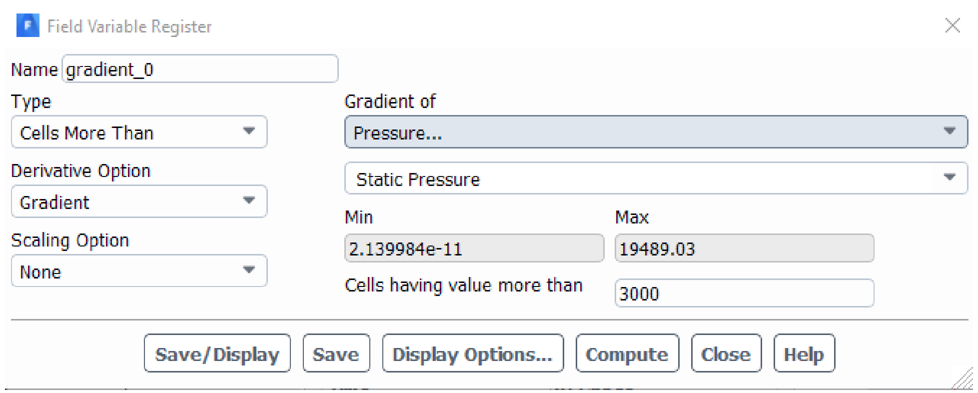


Mach number distribution after Hybrid initialization

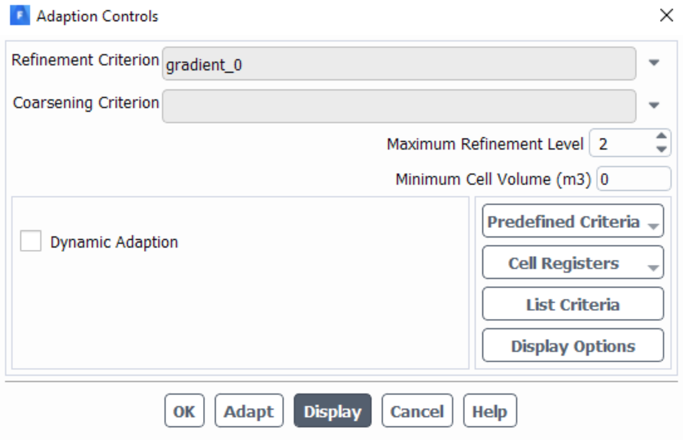
Mach number distribution after FMG initialization

Mesh Adaption

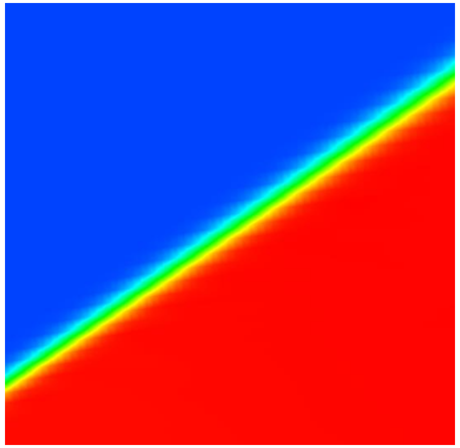
- The gradients across the shock are large and requires higher mesh resolution for resolving these. We use ‘Mesh Adaption’ for this.
- The obtained shock wave is thinner, sharper, and has a better resolution.
- We do this in two steps:
  - Cell Register: we create a cell register using the gradient derivative option for the static pressure in order to identify the cells in the shock region.
  - Refinement: Fluent refines the region identified by the created cell register.



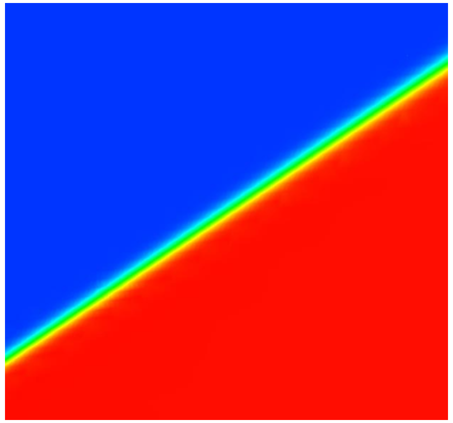
Field variable Register



Adaption Controls



Initial Mesh



Refinement

Ansys