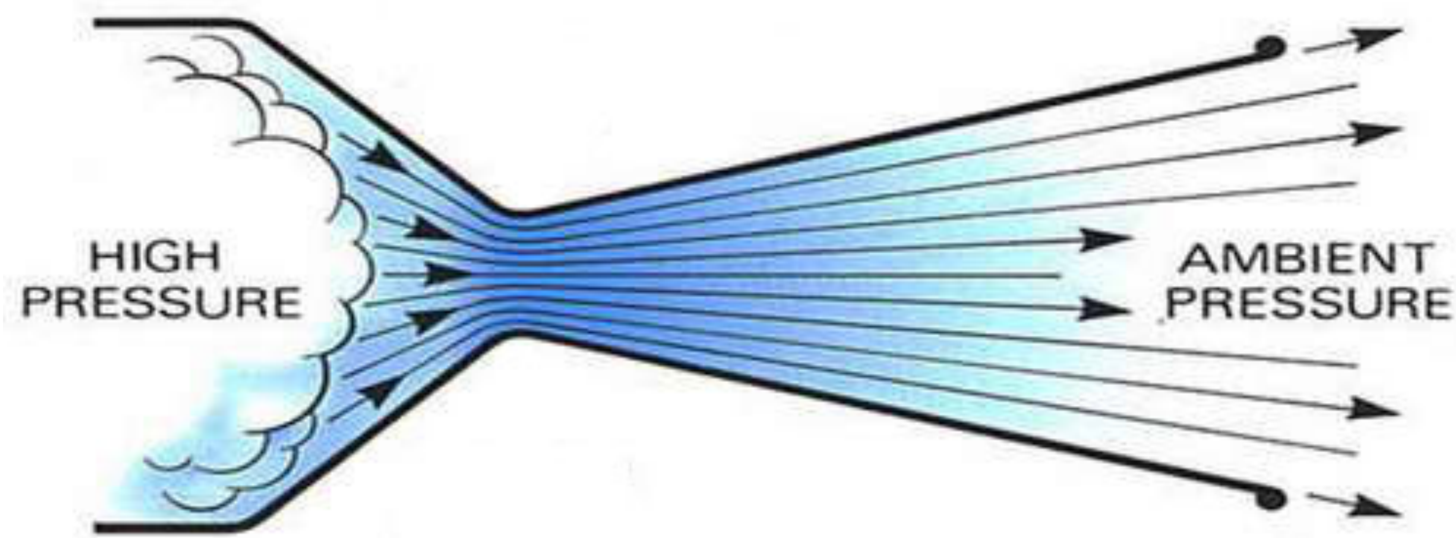
CFD Analysis of a Super Sonic Nozzle

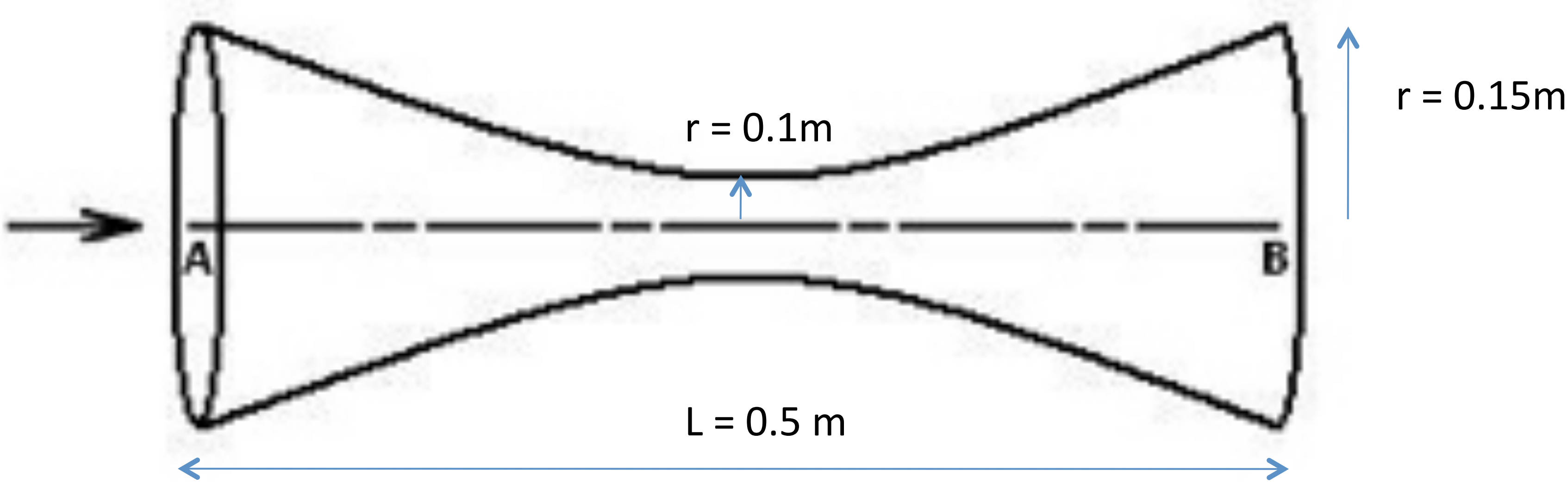


By Max Plomer

Problem Specification

- Air flows at a high speed through a circular cross-section, custom designed converging-diverging nozzle
- Air at inlet is atmospheric: 101,325 Pa and 300K
- Outlet conditions are 3000 Pa and 300K
- Large Reynolds number, viscous effects will be small, therefore flow modeled as invicid

Initial Arbitrary Nozzle Design



- Need an equation for the line of form: $Y = A*X^2 + B$

With points (0 , 0.1) and (0.25 , 0.15)

Therefore: $Y = 0.8*X^2 + 0.1$

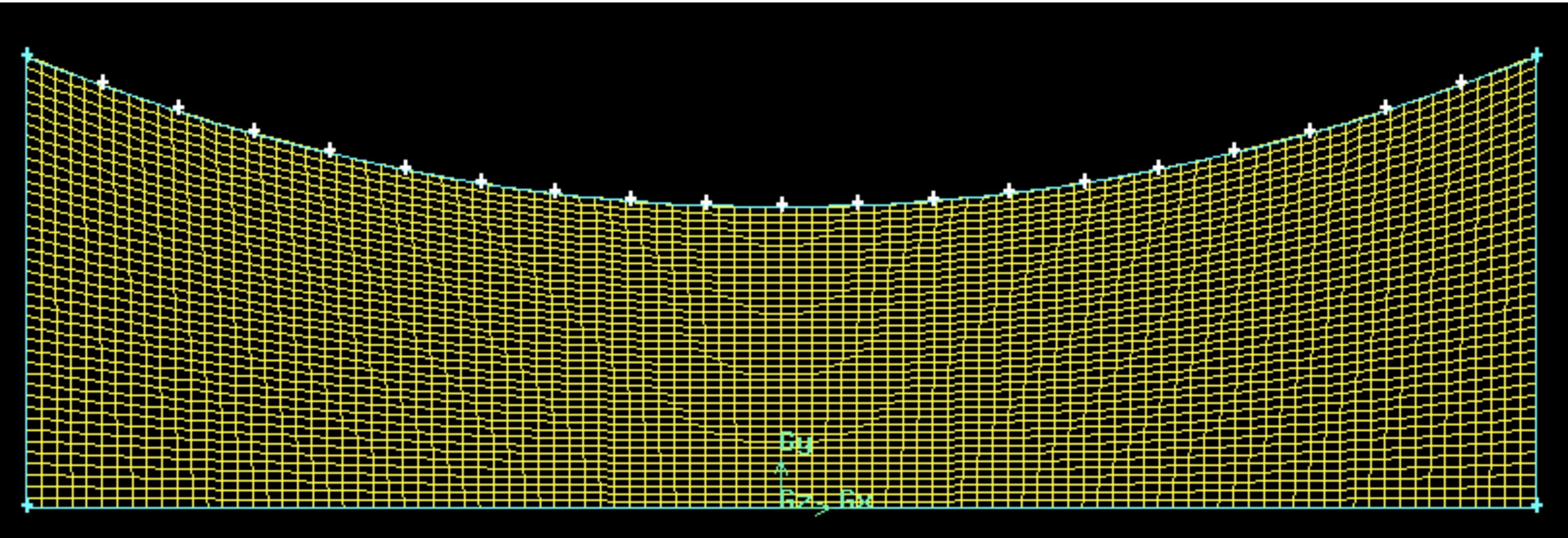
- Later, I increase the outlet radius up until r = 0.3m to see how the mach number at the wall and axis is affected.

Why Varying Outlet Area?

- In Rocket engines a converging-diverging nozzle is used. An increase in the area of the outlet gives a higher mach # at the exit, but changes the range of back pressures in which the rocket can operate.
- With a correct integration of the chemistry for methane or hydrogen, an engineer can design their rocket to be most efficient for all the back pressures of the application.

Mesh

- Since Axi-symmetric, modeled top half only
- Used a 40x100 mesh
- Set boundary types as: Pressure-inlet, Pressure-outlet, Wall & Axis

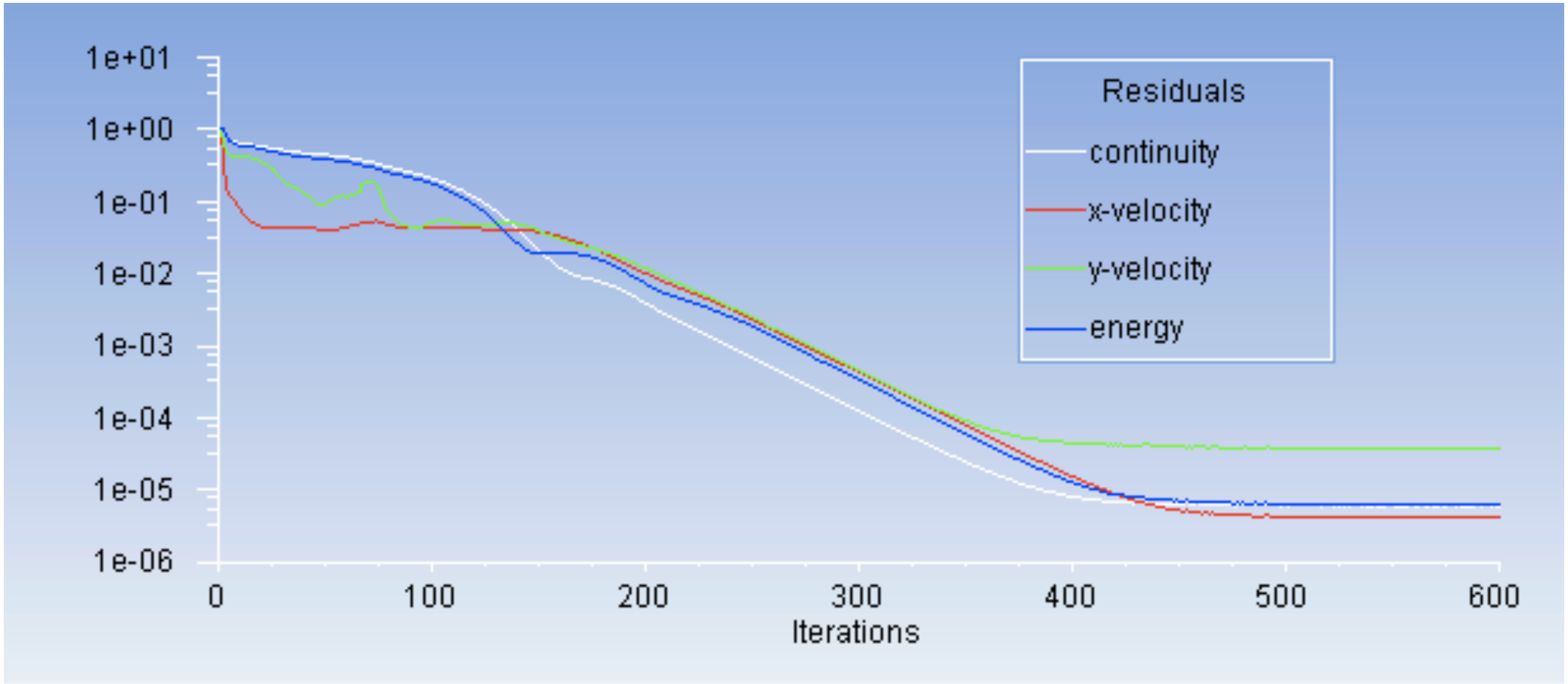


Fluent Physics Options

- Solver: Since we have a high-speed compressible flow, using the density-based solver
- Space: Solve the axi-symmetric form of the governing equations
- Viscous Model: Inviscid, solver will neglect viscous terms
- Using Energy Equation, because since a compressible flow, energy equation will be coupled to continuity and momentum equations
- Density of Air: Ideal gas
- Pressure inlet: 101,325 Pa @ 300K, Pressure outlet: 3,000 Pa@300K
- Solution Methods: Second Order Upwind
- Convergence criteria for : $1*10^{-6}$
- 600 Iterations

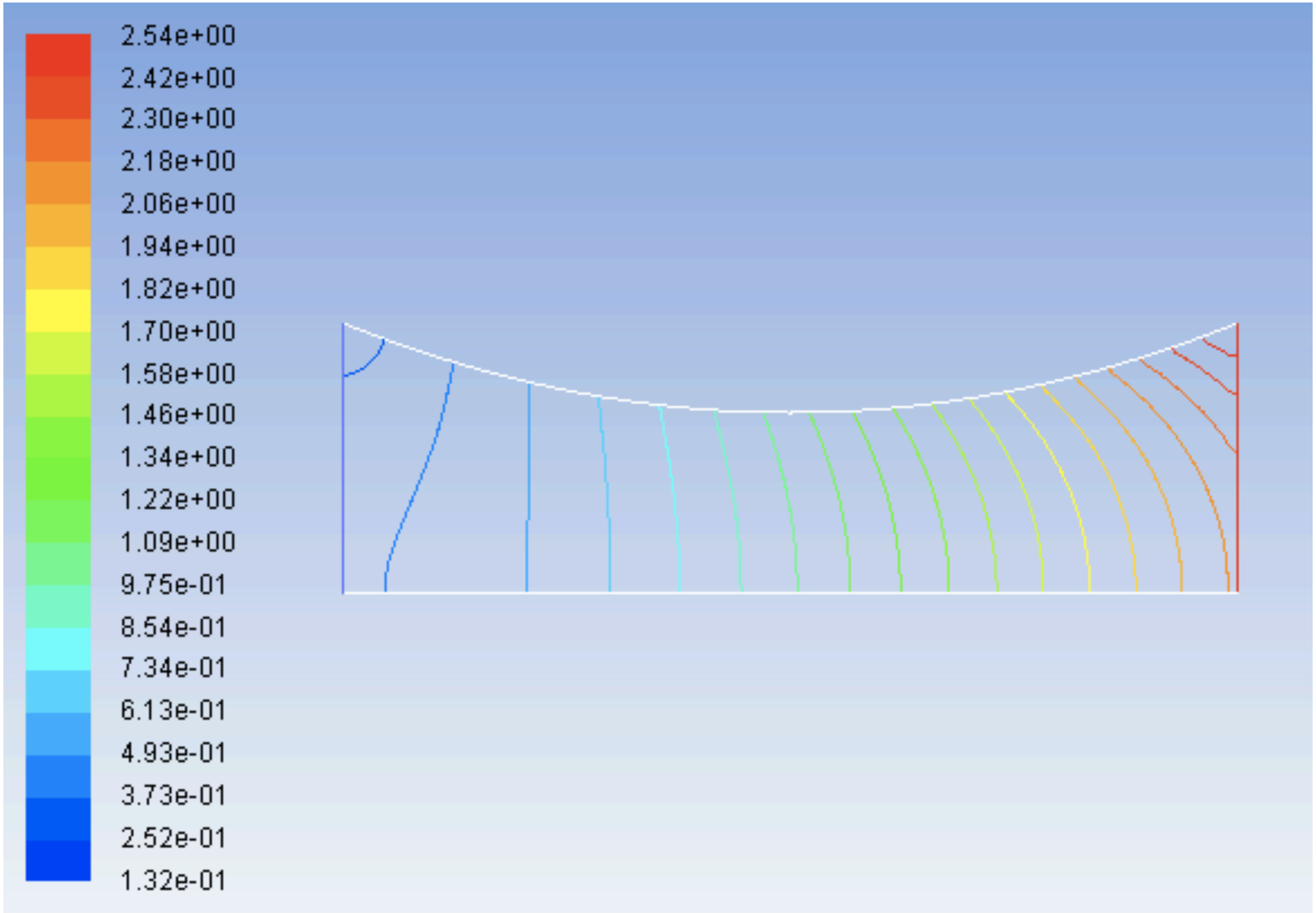
Convergence

- For the given mesh & parameters simulation converges at around 500 iterations



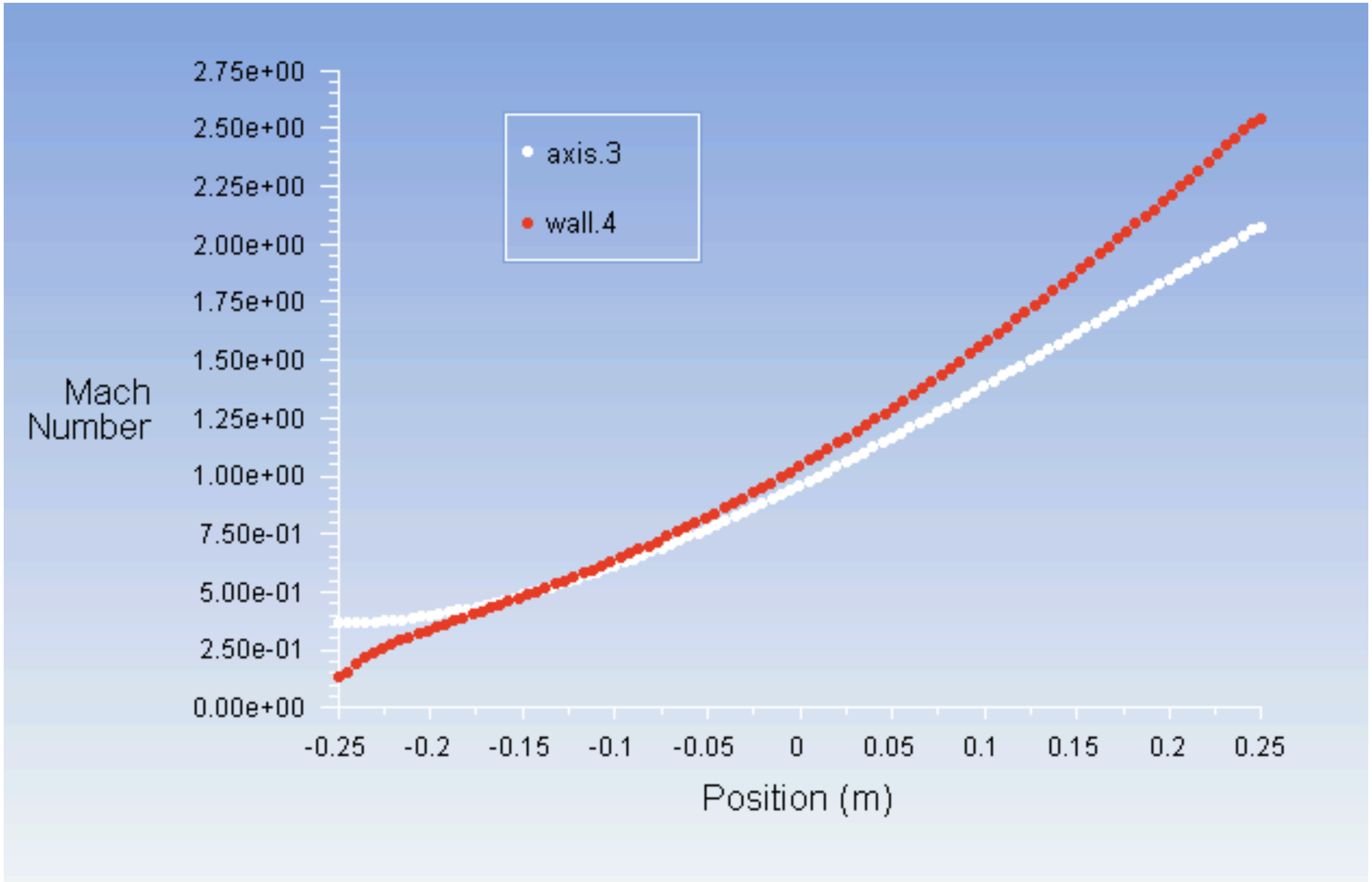
Results – Velocity Contour (Mach #)

- Notice: velocity greater at wall for diverging section



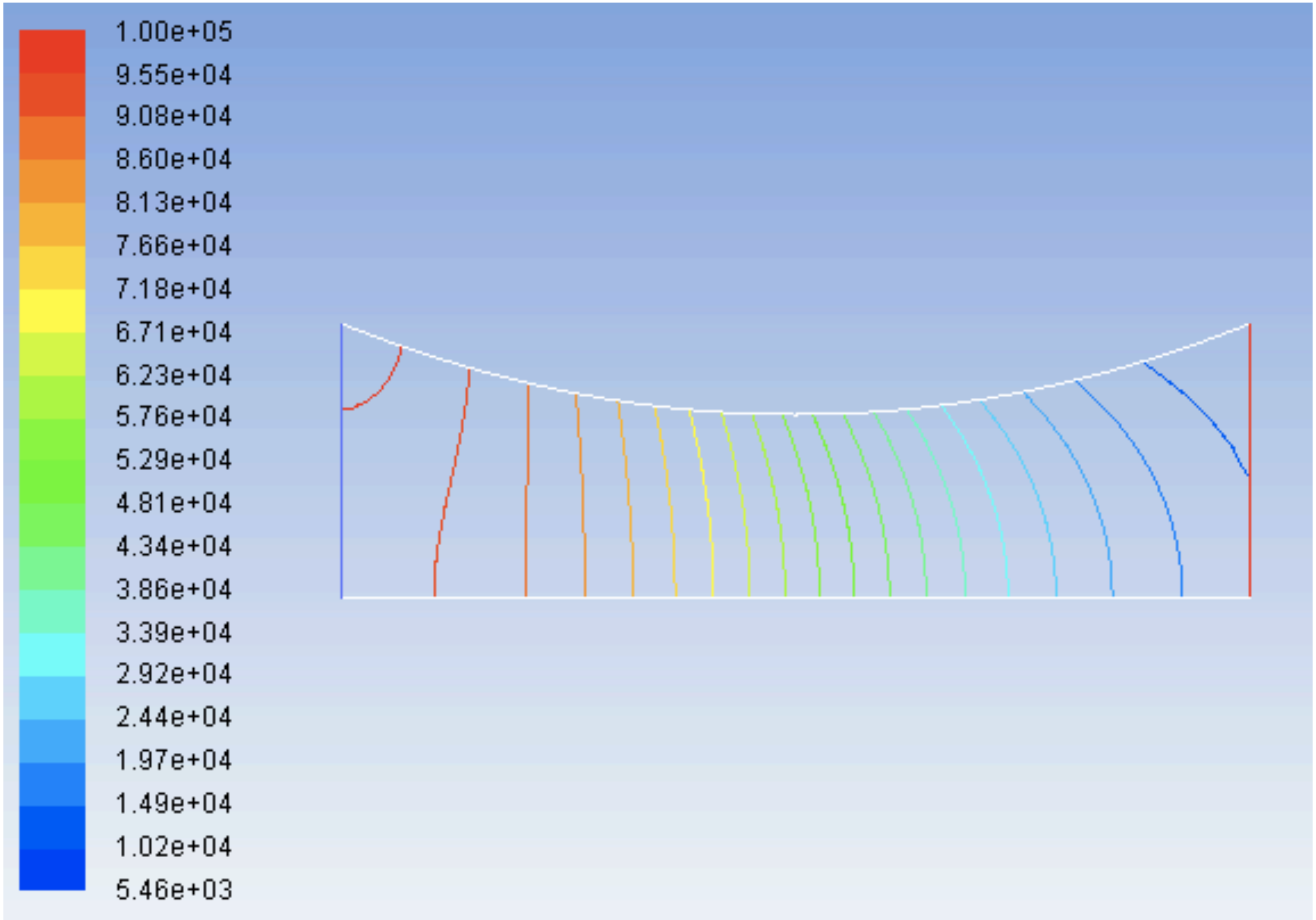
Results – Velocity X-Y Plot

Axis reaches mach 2.05, while Wall reaches mach 2.55



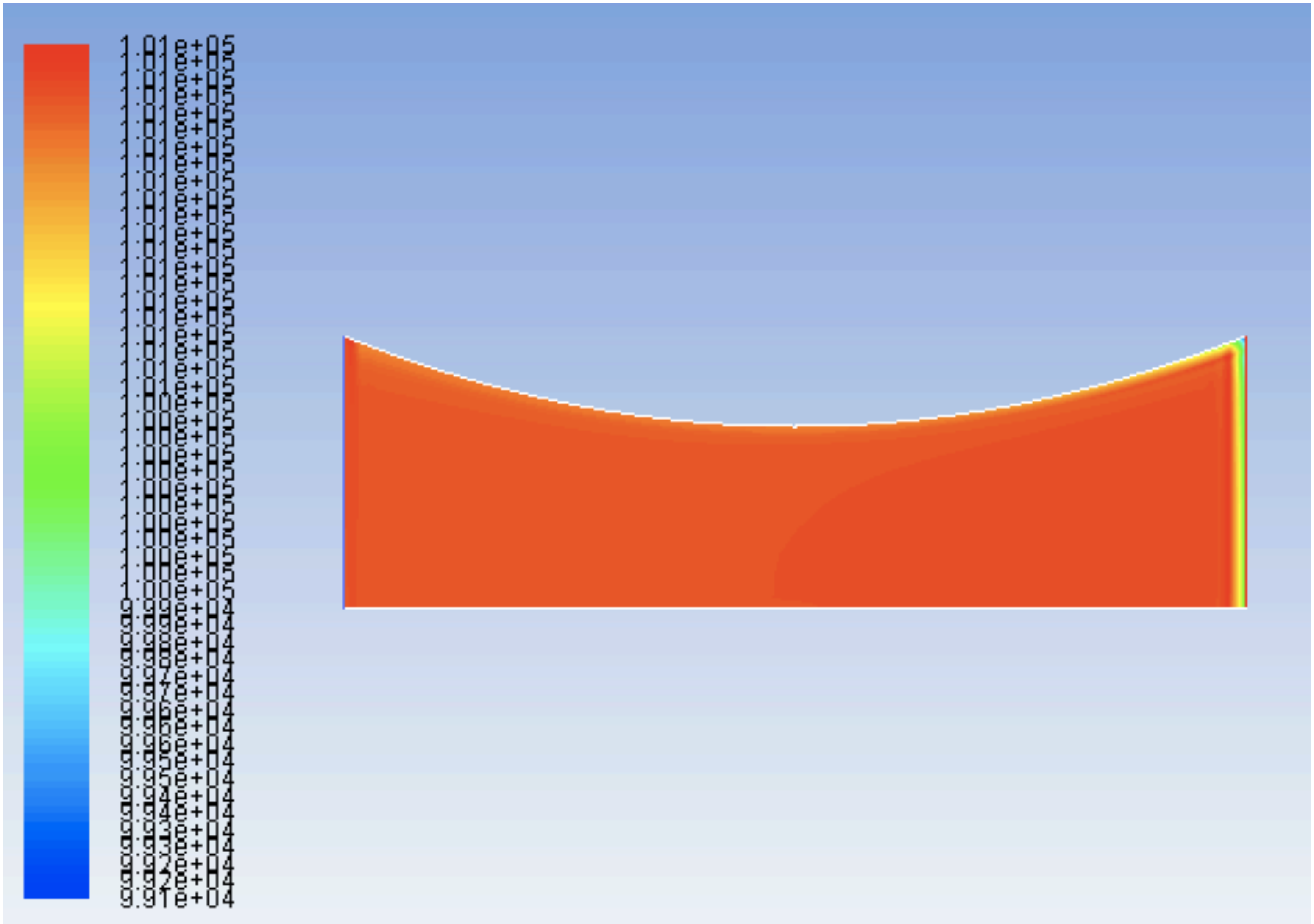
Results – Static Pressure (Pa)

- Notice that pressure decreases from left to right

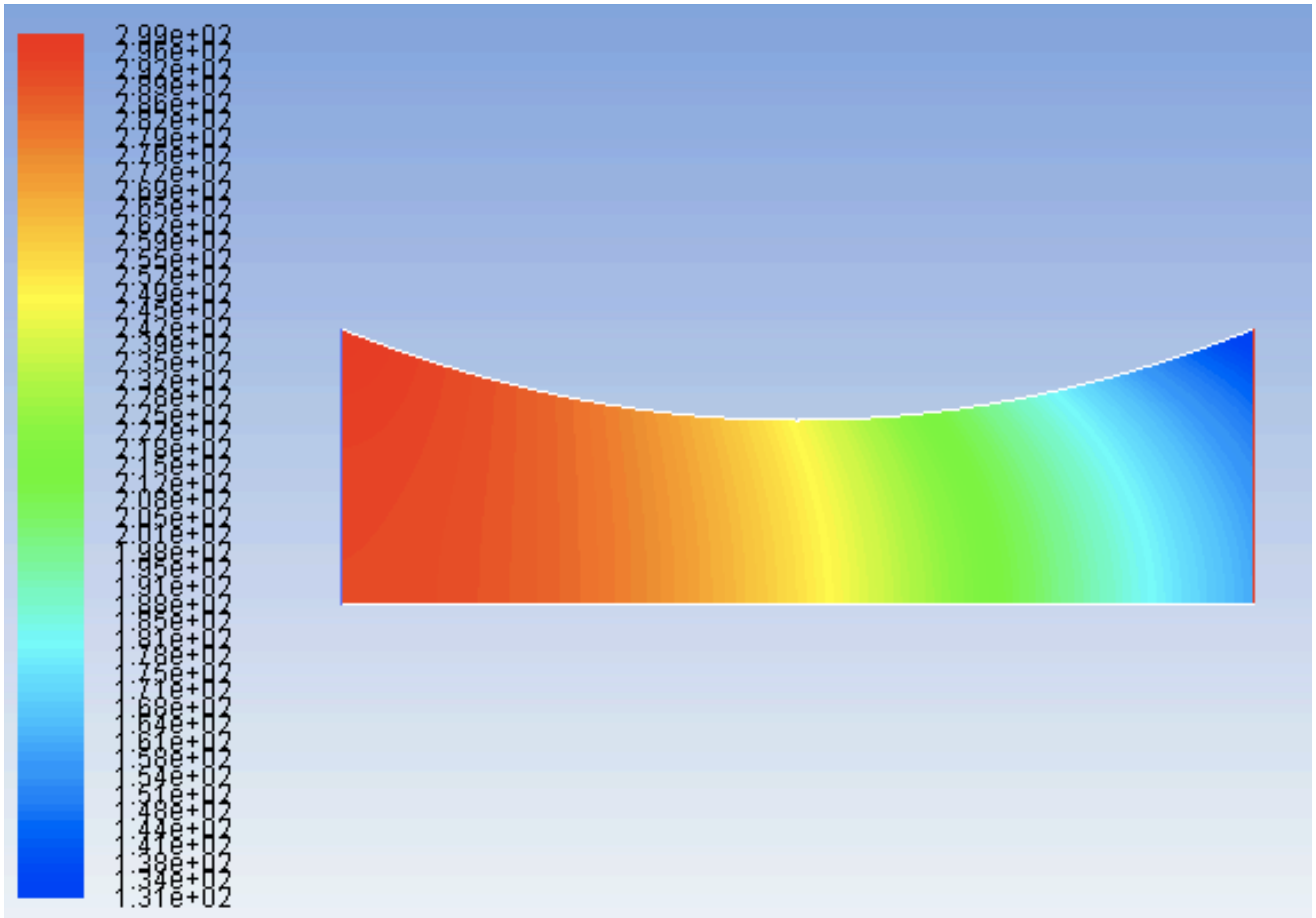


Results – Total Pressure

- Notice pressure loss at outlet, due to false diffusion

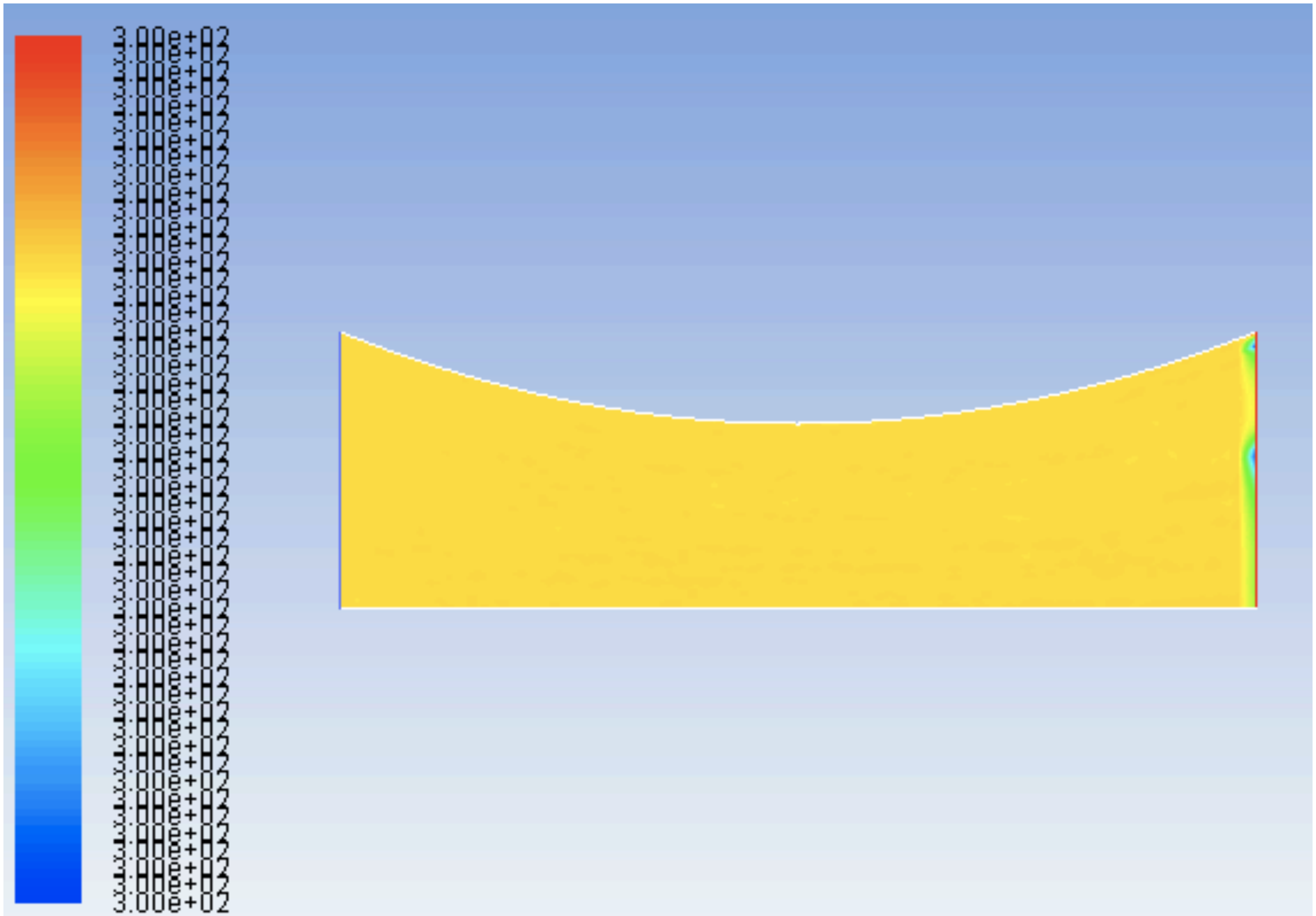


Results – Static Temperature (K)



Results – Total Temperature (K)

- Notice the constant total Temp of 300K, and round off error near the outlet



Varying of Outlet Radius, to see effect on Mach

- Running simulations for R_{outlet} = 0.15m, 0.20m, 0.25m & 0.30m, with all other parameters the same.
- Equation for points left of origin still $Y= 0.8*X^2 + 0.1$
- For right of origin equations are:

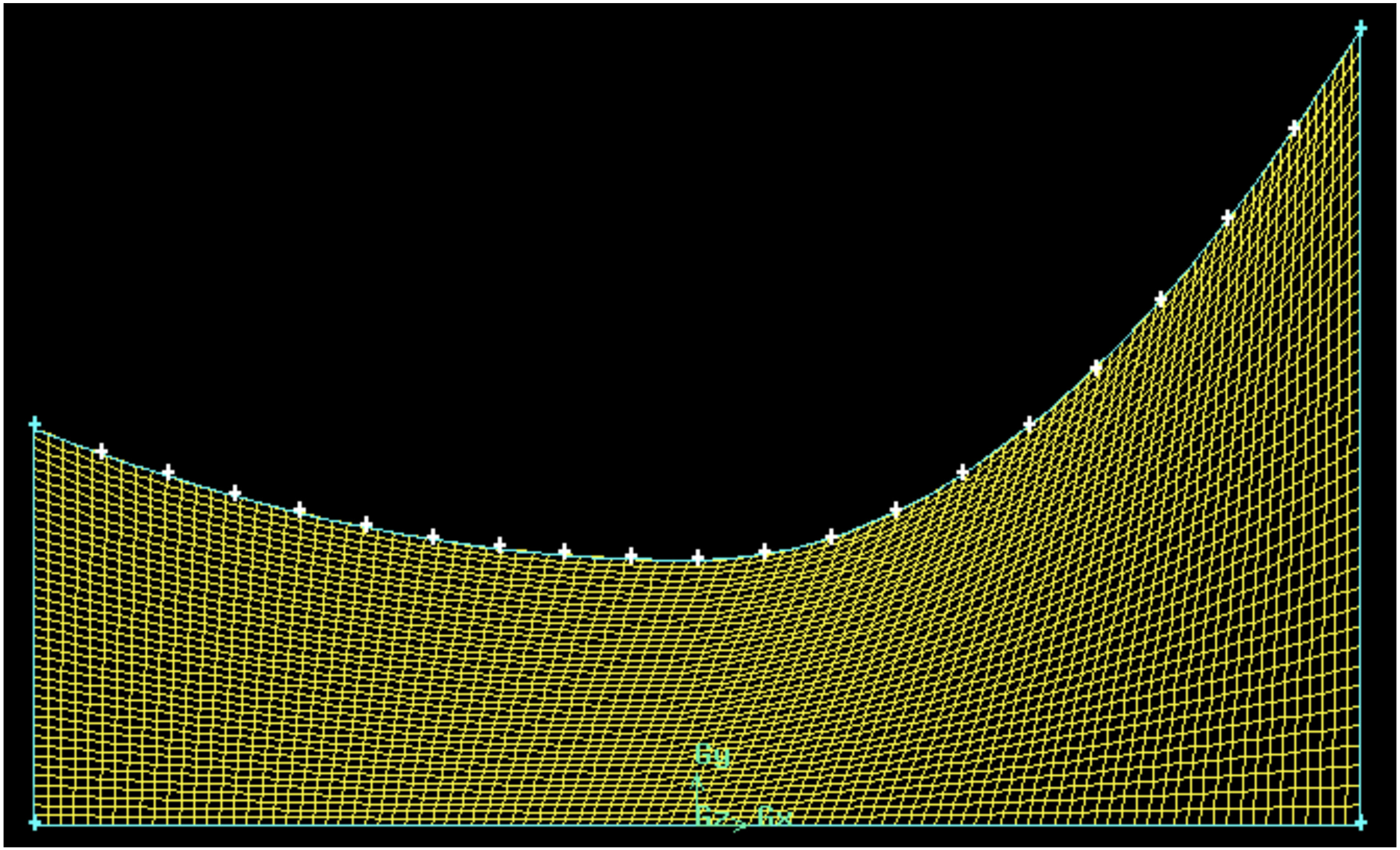
  0.15m: $Y= 0.8*X^2 + 0.1$

  0.20m: $Y= 1.6*X^2 + 0.1$

  0.25m: $Y= 2.4*X^2 + 0.1$

  0.30m: $Y= 3.2*X^2 + 0.1$

Mesh for R_{outlet} = 0.30m



R_{outlet} vs Mach # of Axis & Wall

- R_{outlet} =.15m

Axis mach 2.05, Wall mach 2.55

- R_{outlet} =.20m

Axis mach, Wall mach

- R_{outlet} =.25m

Axis mach, Wall mach

- R_{outlet} =.30m

Axis mach 2.55, Wall mach 5.25