

Inviscid flow over a bump - Optional

The task is to compute the stationary ($\frac{\partial}{\partial t} = 0$), laminar flow field (by solving Euler equations) on the upper half of a 4.2 % thick circular arc “bump on a wall” over the transonic speed range, $M_\infty = 0.5, 0.85$ and 1.65 . The results for the $M_\infty = 0.85$ should be compared, at least qualitatively, to literature results reproduced in the handout (Rizzi, Viviands, Eds, *Numerical Methods for the Computation of Inviscid Transonic Flows with Shock Waves*, Notes on Numerical Fluid Mechanics, vol3, Vieweg, 1981).

A MATLAB code that solves the Euler equations, see below, by the MacCormack scheme, can be downloaded from the homepage of the course (Nadas part). In the directory `Navier_Stokes` you find the codes related to solving the Euler equations and in the directory `mesh` you find the files necessary to create the mesh.

The Euler equations in 2D are:

$$\mathbf{U}_t + \mathbf{F}(\mathbf{U})_x + \mathbf{G}(\mathbf{U})_y = 0$$

where

$$\mathbf{U} = \begin{pmatrix} \rho \\ \rho u \\ \rho v \\ E \end{pmatrix} \quad \mathbf{F}(\mathbf{U}) = \begin{pmatrix} \rho u \\ \rho u^2 + p \\ \rho uv \\ (E + p)u \end{pmatrix} \quad \mathbf{G}(\mathbf{U}) = \begin{pmatrix} \rho v \\ \rho uv \\ \rho v^2 + p \\ (E + p)v \end{pmatrix}$$

Here, ρ is the density, u is the velocity in the x-direction, v is the velocity in y-direction and $E = \rho(e + (u^2 + v^2)/2)$, where e is the internal energy.

The boundary conditions are set as follows:

- Ingoing characteristics: the variables are set
- Outgoing characteristics: extrapolation from the inner of the domain.

At the inflow (left) and outflow (right): In the supersonic case, the variables are extrapolated and in the subsonic case, the Riemann invariants are extrapolated.

At the top: Riemann invariants are extrapolated

At the bottom (wall curvature is neglected in pressure extrapolation to wall): $(u, v)^T \mathbf{n} = 0$, $\partial \rho / \partial n = 0$ and $\partial E / \partial n = 0$. Here n denotes the normal to the wall.

You run the program by `NS.m` which is the main file. Before you run the program you have to create a mesh. This is done by running the mesh generator file `ttranf.m` in the `mesh` directory. In this file you have to specify the number of gridpoints and the stretching of the mesh, see last page (copy of `ttranf.m`). A file `NAVIER_FVM.MESH` is created containing the information about the mesh which is used by the code `NS`. The mesh should be stretched so that the cells at the bump corners are approximately square.

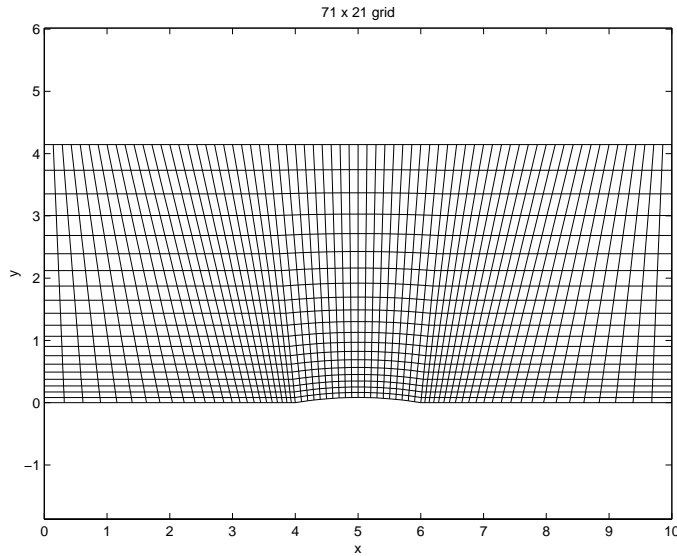


Figure 1: Mesh

Start with a grid with about seventy cells in the horizontal direction, with twenty on the foil, and twenty cells in the vertical direction. The mesh is stretched to improve the resolution at the leading (front) and the trailing (back) edge where the flow gradients are expected to be greatest. The mesh generator file has been prepared for this. Note, you may have to change the mesh depending on the flow case.

The program is run by `NS.m`, and the case set up runs 100 steps for $M_\infty = 0.5$ and plots the Mach number and density residuals, $\|\rho^{n+1} - \rho^n\|_2$ and $\|\rho^{n+1} - \rho^n\|_\infty$. The Courant number, $CFL = 0.8$ and the artificial viscosity parameters $EPS2 = 0.25$ and $EPS4 = 0.004$ in both x and y directions, which are standard settings for transonic flow. Note that you might have to change these parameters in order to obtain a reasonable solution.

Your tasks are the following:

1. Run the three cases ($M_\infty = 0.5, 0.85$ and 1.65) on a *medium* grid like 70×20 grid points. For the $M_\infty = 1.65$ you might need a grid with better resolution. Be sure to run enough timesteps (of order 500-3000) to reach reasonable convergence (stationary solution). You should have a decrease of the density residual of at least three orders of magnitude.

For each case:

- Compute plot the surface pressure coefficient, (C_p vs x at $y = 0$). The pressure coefficient is given by

$$C_p(x, y) = \frac{p(x, y) - p_\infty}{\frac{1}{2}\rho_\infty U_\infty^2}$$

- Show plots of the iso-lines of the Mach number vs (x, y) .

For the $M_\infty = 0.85$ case compare your results to the results reported in Rizzi and Viviand.

2. Grid assessment

The level of artificial dissipation of the scheme, especially the solid wall boundary conditions, can be assessed by plots of entropy, S , and stagnation enthalpy, H .

$$S = \frac{p}{\rho^\gamma} \quad H = \frac{E + p}{\rho}$$

For inviscid flow:

- entropy is constant along stream lines, except at shocks
- enthalpy is constant along stream lines, also at shocks.

Spurious enthalpy production comes from the solid wall, especially edges, grid line slope discontinuities, and grid cell size changes. Thus, an enthalpy plot can be used to assess grid quality.

Entropy changes can arise at the shock (correct) and (spurious) at leading and trailing edges of the bump. It follows from this that the grid should be clustered at edges.

Compute and plot the entropy and the enthalpy. Does the entropy increase over the shock ?

Try a mesh with a stronger clustering at the corners of the bump. Does this affect the enthalpy and/or the entropy. Check and report the results.

3. Accuracy assessment

Run one of the cases with a coarser and a finer grid. Compare the solutions and report your result.

For all the cases you have to experiment with the artificial viscosity parameters to find “optimal” ones. Note that too large values may give parabolic instabilities since the code only checks the CFL conditions related to the wave speeds (characteristics).