BI3 - Inviscid bow shock upstream of a blunt body in supersonic flow

Contact: Scott M. Murman, Scott.M.Murman@nasa.gov

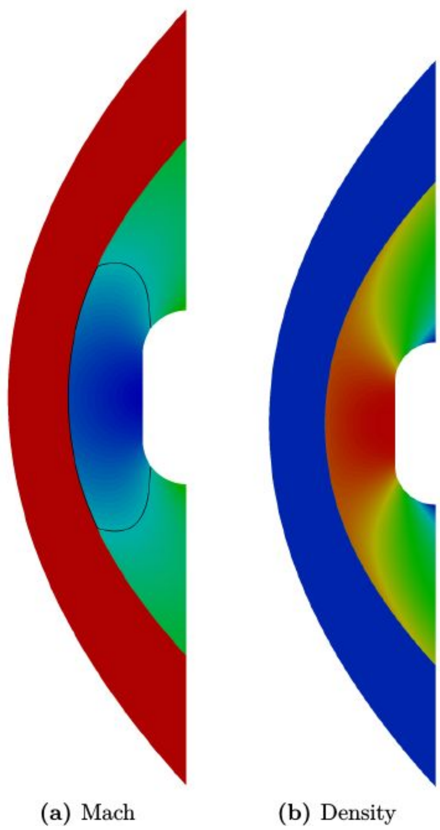


Figure 1: Computed flowfield. Black line shows the Mach 1 contour.

Overview

This case is designed to isolate testing of the shock-capturing properties of schemes using the detached bow shock upstream of a two-dimensional blunt body in supersonic flow. This case is computationally expedient, being steady, two-dimensional, inviscid flow, with well-defined boundary conditions.

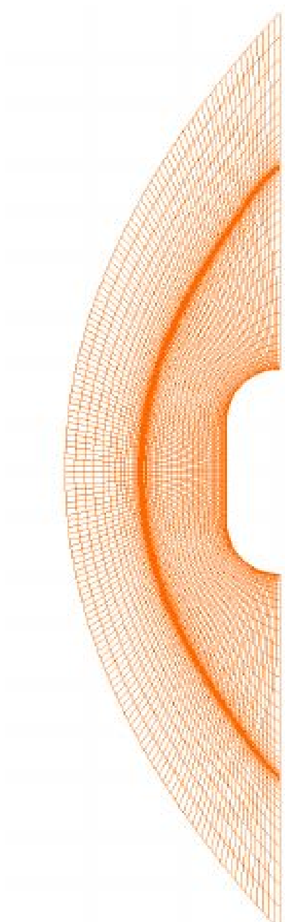
The geometry is a flat center section, with two constant radius sections top and bottom (cf. Fig. 1). The flat section is one unit length, and each radius is 1/2 unit length. While the flow is symmetric top and bottom, a full domain is computed to support potentially spurious behavior.
The aft section of the body is not included to avoid developing an unsteady wake.

In steady inviscid flow the total enthalpy, H = (ρE + p)/ρ, is constant, where ρE is the total energy. The error in this quantity provides a first quantifiable measure of the quality of the computed solution of the general Euler equations (as opposed to schemes which specifically optimize for steady, inviscid flow and enforce H = const.). Along the stagnation streamline, the stagnation pressure on the cylinder surface is predicted by the Rayleigh-pitot formula,

$$\frac{p_{02}}{p1} = \frac{1-\gamma+2\gamma M_1^2}{\gamma+1} \cdot \left(\frac{(\gamma+1)^2 M_1^2}{4\gamma M_1^2 - 2(\gamma-1)} \right)^{\frac{\gamma}{\gamma-1}}$$

where subscript 1 refers to conditions upstream of the shock and 2 to the stagnation point. This provides a second quantity to assess the accuracy of schemes, and one which is directly related to engineering utility in supersonic flow simulations.

Computational Mesh



A series of computational meshes are provided which are progressively adapted to the “infinite resolution” solution computed using the standard Overflow 2nd-order central-differencing scheme with 2nd- and 4th-order dissipation blended using a pressure sensor. These meshes are not hierarchical. At each refinement the clustering near the shock location and the surface in increased. These meshes all cluster around the asymptotic shock location (i.e. at coarse resolutions the computed shock will be in the incorrect location relative to the mesh, but should converge to the predicted location). The mesh is designed so that a single element/cell/stencil straddles the asymptotic shock location.

Meshes are provided in CGNS format (grid0.cgns, grid1.cgns, ...) for FD/FV solvers, and as a set of GMSH meshes for DG, SD and FR solvers (N2/grid0.msh, N2/grid1.msh, ...), classified by the formal accuracy of the code/mesh N = p + 1 (i.e. N = 2 uses p = 1 linear elements). It is expected that the full series of meshes from 0 to 4 will be run.

Boundary Conditions

The inflow and outflow are both supersonic, so Dirichlet and Neumann boundary conditions respectively are prescribed. The incoming freestream is at Mach 4. The solid surface uses a standard impermeable wall specification (u · n = 0).

Deliverables

In order to provide consistent predictions it is important that all simulations converge to as close to machine epsilon as possible. This is likely to be difficult for many schemes, e.g. due to the carbuncle phenomena, or limiter oscillations. Entries should provide convergence of the residual over all dependent variables for each case, along with computed values of the rms error in total enthalpy as well as the pointwise error in stagnation pressure at the stagnation point, as functions of the cost in terms of tau work units. Further quantities may be requested to provide better understanding the relative behavior of schemes.

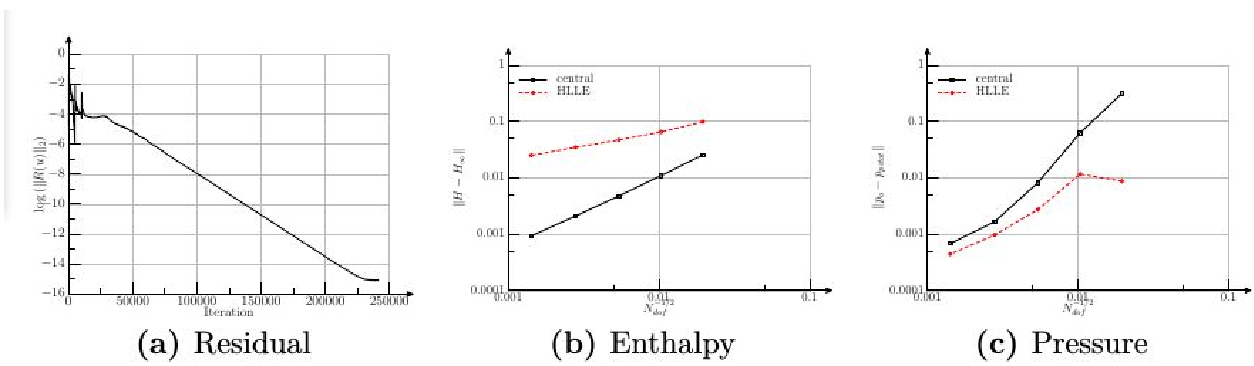


Figure 3: sample convergence results using the overflow solver.

Defining H_{ref} as the freestream/reference value of the total enthalpy, its RMS or L2 error is formally defined as

$$Err_H = \sqrt{\frac{\sum_i^N (H_i - H_{ref})^2}{N}}$$

with the index i running over all degrees of freedom for point based methods such as FVM and FDM, and as

$$Err_H = \sqrt{\frac{\int_\Omega (H - H_{ref})^2 dV}{\int_\Omega dV}}$$

for finite element like methods such as CG, DG, FR, ...

Figures 3a-c present sample results using the Overflow solver for the 2nd-order central differencing scheme described above, and the 3rd-order MUSCL scheme using the HLLE flux and van Albada limiter.