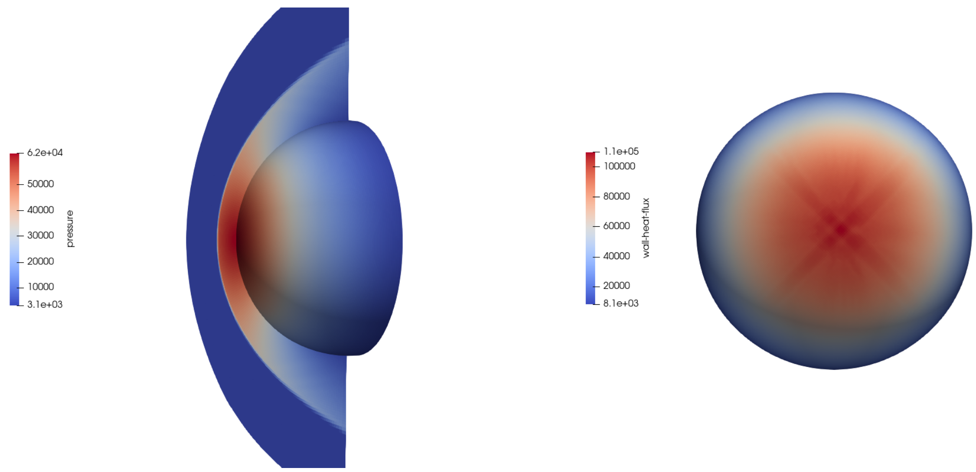
High Fidelity CFD Workshop 2021:
High Speed Steady Advanced Case: Blottner Sphere

Travis Fisher (tcfishe@sandia.gov)

February 11, 2020



(a) Pressure (b) Heat Flux

Figure 1: Example flow results are shown for the Blottner sphere.

1 Summary

The purpose of this case is to evaluate the ability of solvers to correctly and efficiently predict heating on the surface of a sphere in a three-dimensional high-speed flow. Typical issues encountered by solvers include poor predictions due to excessive dissipation in the boundary layer, carbuncles, and mesh-imprinted solutions. The reference case was simulated by Blottner (1990) using an axisymmetric solver, which naturally suppresses some of the issues observed in three-dimensional calculations. For the workshop, we will solve this case using a three-dimensional solver. All participants must use one set of the provided meshes. In addition, adaptive mesh results will be accepted.

2 Geometry, Governing Equations, and Flow Conditions

2.1 Geometry

The provided meshes use a different sphere radius than the original Blottner paper. The radius is $r = 0.0635\,\text{m}$. The geometry is shown in Figure 2.

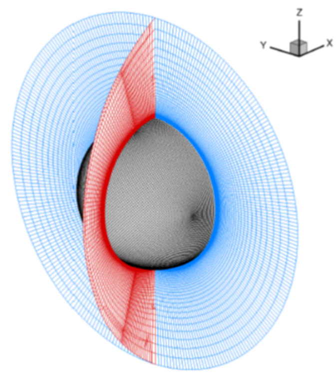


Figure 2: The geometry and grid topology for the Blottner sphere problem is shown.

2.2 Governing Equations

The compressible Navier-Stokes equations should be simulated using the perfect gas assumption (e.g. single species, constant specific heat) and Sutherland's Law for viscosity.

2.3 Flow conditions

The case is summarized in Table 1. Note that the wall temperature ratio is with respect to the static freestream temperature, not the stagnation temperature.

Table 1: The flow conditions for the Blottner sphere case.

Specific Heat Ratio, γ	1.4
Mach Number, Ma	5.0
Reynolds Number, Re_D	1.8875×10^6
Prandtl Number, Pr	0.72
Wall Temperature Ratio, T_w/T_∞	1.308

3 Meshes

Three sets of meshes are provided:

1. Structured multi-block meshes,
2. Unstructured hexahedral meshes, including high-order curved meshes,
3. Unstructured tetrahedral meshes, including high-order curved meshes.

The mesh topology avoids the need to handle singular or degenerate elements.

4 Boundary Conditions

The inflow and outflow boundaries can use whatever boundary condition enforcement is appropriate for the solver. It is unlikely that the choice will impact the computed metrics, but it may impact the robustness of certain solvers depending on numerical sensitivity. We note that due to the truncated domain, it will be nearly impossible to get convergent behavior at the outflow boundary in the boundary layer.

We recommend the solver switch to a subsonic back pressure condition ten times lower than the freestream static pressure when the flow goes subsonic in the outflow. We also recommend a zero normal viscous flux.

The wall boundary condition is an isothermal, no-slip condition with a wall temperature ratio,

$$\frac{T_w}{T_\infty} = 1.308, \tag{1}$$

where T_w is the isothermal wall temperature and T_∞ is the static freestream temperature.

5 Initial Conditions

Since this is a steady problem, no initial conditions are required. However, as part of the required submittal, the approach for defining the initial condition should be described. For example, if the solver used creates an artificial boundary layer, please give a brief description of the approach. If a precursor simulation is used, please include the details and also add the cost of the precursor simulation to the total work units.

6 Outputs

We would like to evaluate six different outputs.

1. Stagnation point wall heat flux for each mesh,

$$q_w = -\kappa \frac{\partial T}{\partial x_i} n_i \bigg|_{y=0,z=0}, \tag{2}$$

where κ is the thermal conductivity, T is the temperature, and n is the outward facing wall normal.

2. Integrated wall heat flux for each mesh,

$$Q = \int_{\Gamma_{wall}} -\kappa \frac{\partial T}{\partial x_i} n_i dS, \tag{3}$$

where Γ_{wall} is the entire wall boundary. Alternate approaches to computing wall heat flux, such as using the viscous flux component corresponding to the energy equation, are acceptable.

3. Wall heat flux profiles for each mesh:
   (a) along the intersection of the wall and the y-z plane,
   (b) along the intersection of the wall and a $\pi/4$ rotation from the y-z plane.

4. Static pressure profile for each mesh:

   (a) along the intersection of the wall and the y-z plane,
   (b) along the intersection of the wall and a $\pi/4$ rotation from the y-z plane.

5. Static pressure profile along the stagnation streamline for each mesh.

6. Static temperature profile along the stagnation streamline for each mesh.

Pressure and temperature should be non-dimensionalized by p_∞ and T_∞, respectively. Stagnation heat flux should be nondimensionalized by $\frac{\kappa_\infty T_\infty}{r}$ and integrated heat flux should be nondimensionalized by $\frac{\kappa_\infty T_\infty}{2\pi r^3}$, where κ_∞ corresponds to th freestream thermal conductivity used in the solver.

7 Requirements

Each submittal should consist of the following information:

1. Three meshes from one of the provided sets should be simulated. Poor results are still informative data to the community. All submissions are welcome.

2. The nonlinear residual convergence of the solver is provided for each mesh. Residuals can be absolute or relative and can be per equation or aggregated.

3. The number of work units to reach the converged solution is provided for each mesh.

4. The six metrics described in Section 6 are provided in a simple text format.
   (a) The profiles and nonlinear convergence should be provided in a comma separated variable or similar format. (Readable by numpy *loadtxt*)
   (b) The scalar quantities should be tabulated in a text file similar to Table 2.

Table 2: Example results for Blottner Sphere scalar quantities.

Mesh	Work Units	Stagnation Heat Flux	Integrated Heat Flux
Structured c2	-	-	-
Structured c1	-	-	-
Structured c0	-	-	-

8 Modifications/Adaptivity

All adaptive approaches are welcome including mesh refinement and shock fitting. However, if possible you must still submit results on one set of the provided meshes. Exceptions will be considered for solvers that require adaptivity for robustness.

In addition to the required metrics in Section 7, adapted cases should provide aggregate work units required to perform the adaptation and the final number of solution points used in the simulation.

Acknowledgements

Derek Dinzl and Micah Howard helped in defining this case, and the other case organizers for the workshop provided valuable feedback. Steve Karman has generated high-order curved meshes.

References

Blottner, Frederick G. 1990 Accurate Navier-Stokes results for the hypersonic flow over a spherical nosetip. *Journal of Spacecraft and Rockets* **27** (2), 113–122.