High Fidelity CFD Workshop 2021:
High Speed Steady Challenge Case: Hi-Fire 1

Travis Fisher (tcfishe@sandia.gov)

February 11, 2020

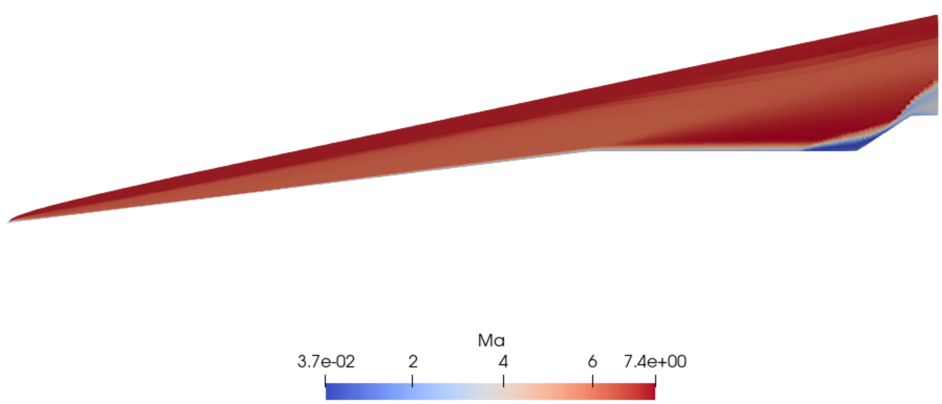

Figure 1: Example results are shown for flow on the Hi-Fire 1 geometry.

1 Summary

The purpose of this case is to evaluate the ability of solvers to predict converged heating on a simple gemoetry in high-speed flow at angle of attack. The Hi-Fire 1 geometry is a blunt sphere-cone with a cylindrical section and a flare. Wadhams *et al.* (2008) The flow conditions correspond to a Mach 7.2 flow with a 2° angle-of-attack. Although the flow conditions correspond to a ground test experiment, the purpose of this workshop case is to evaluate the ability of different flow solvers to converge to the same solution with mesh refinement. Thus, comparison to experiment is outside the scope of the workshop case.

2 Geometry, Governing Equations, and Flow Conditions

2.1 Geometry

The geometry is fully described in Wadhams *et al.* (2008). An image of the wall surface is shown in Figure 2.

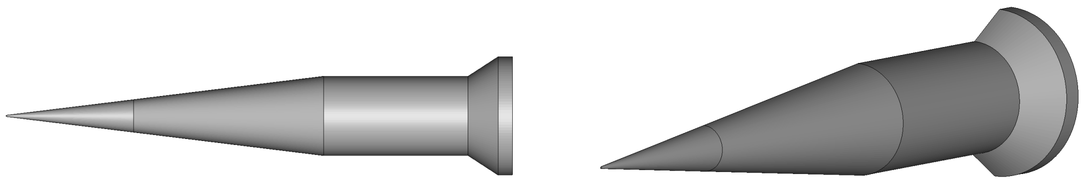

Figure 2: The geometry for the Hi-Fire 1 problem is shown.

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. All submissions should include results using the negative Spalart Allmaras turbulence model. Allmaras *et al.* (2012) Participants should also run without a turbulence model to evaluate the convergence of the heat flux on the conical section. We will allow for additional turbulence model submissions that showcase alternate convegence behavior. This is described in Section 8.

2.3 Flow conditions

The case is most similar to the Run 34 in Wadhams *et al.* (2008) and is summarized in Table 1. Note that the wall temperature ratio is with respect to the static freestream temperature, not the stagnation temperature.

The provided meshes have the body aligned with the Cartesian coordinate directions. The angle-of-attack must be imposed through boundary conditions.

Table 1: The flow conditions for the Hi-Fire 1 case.

Specific Heat Ratio, γ	1.4
Mach Number, Ma	7.18
Freestream Reynolds Number, Re_∞	$10.123 \times 10^6\,/m$
Prandtl Number, Pr	0.72
Angle-of-Attack, α	2°
Wall Temperature Ratio, T_w/T_∞	1.279

For the geometry in the provided meshes, the dimensional reference temperature is $T_\infty = 321.91\,K$ and the reference density is $0.070215\,kg/m^3$.

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.

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

$$\frac{T_w}{T_\infty} = 1.279, \tag{1}$$

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. Wall heat flux profiles for each mesh and turbulence model:
   (a) along the intersection of the wall and the y-z plane on the windward side,
   (b) along the intersection of the wall and the y-z plane on the lee side,
   (c) along the intersection of the wall and the x-z plane.
2. Static pressure profile for each mesh and turbulence model:
   (a) along the intersection of the wall and the y-z plane on the windward side,
   (b) along the intersection of the wall and the y-z plane on the lee side,
   (c) along the intersection of the wall and the x-z plane.

Pressure should be non-dimensionalized by p_∞. Heat flux should be nondimensionalized by $\frac{\kappa_\infty T_\infty}{r}$, where κ_∞ corresponds to th freestream thermal conductivity used in the solver and r corresponds to the radius of the cylindrical section. ($r = 0.137275\,m$ in the provided meshes.) Note that for laminar cases the profiles can stop at the end of the cone section.

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 and turbulence model. 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 work units should be tabulated in a text file similar to Table 2.

Table 2: Example results for Hi-Fire 1 summary.

Mesh	Work Units
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.

It is well-known that the Spalart-Allmaras model fails to accurate predict the separation region in this flow, therefore there is great interest in other turbulence models. We welcome results from other turbulence models such as SA-QCR or SST. Emphasis of these results should still be a demonstration of convergence. Please inform the test leaders of your intended alternate model so this can be communicated to the broader group. The more people that run a particular model, the better.

At least one set of SST results will be provided by the test leaders.

Acknowledgements

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

References

ALLMARAS, S. R., JOHNSON, F. T. & SPALART, P. R. 2012 Modifications and clarifications for the implementation of the Spalart-Allmaras turbulence model. In *7th International Conference on Computational Fluid Dynamics*, , vol. ICCFD7-1902. Big Island, Hawaii.

WADHAMS, T. P., MUNDY, E., MACLEAN, M. G. & HOLDEN, M. S. 2008 Ground test studies of the HIFiRE-1 transition experiment part 1: Experimental results. *Journal of Spacecraft and Rockets* **45** (6), 1134–1148.