2nd-Order CESE Results For C1.1: Transonic Ringleb Flow

David J. Friedlander

NASA Glenn Research Center, Cleveland, OH, 44135

1 Code Description

The Conservation Element and Solution Element (CESE) method [1, 2] was used as implemented in the NASA research code ez4d [3]. The CESE method is a time accurate formulation with flux-conservation in both space and time. The method treats the discretized derivatives of space and time identically and utilizes a staggered mesh approach consisting of conservation elements (CE) and solution elements (SE). While high-order versions of the method exist [4, 5, 6], the 2nd-order accurate version was used. In regards to the ez4d code, it is an unstructured Navier-Stokes solver coded in C++ with serial and parallel versions available. As part of its architecture, ez4d has the capability to utilize multi-thread and Messaging Passage Interface (MPI) for parallel runs.

2 Case Summary

Cases were run on a single Intel Xeon W3680 core and considered converged when the L2 density residual dropped ten orders of magnitude from the initial solution. Times for running the TauBench executable ranged from 8.042s to 8.292s with an average of 8.154s.

3 Meshes

Meshes were made from scratch using Pointwise®and tried to mimic the structured versions provided by the workshop. The structured versions of the grids were then diagonalized by cutting each structured cell into two triangular cells using the "best fit" option within Pointwise®. Mesh refinement consisted of doubling the number of cells in the i and j directions per grid level while keeping cell spacing uniform. See Table 1 for structured domain sizes. While inflow and outflow boundary conditions were utilized for the top and bottom of the domains, two different boundary conditions were applied at the left and right boundaries of the domains: an inviscid solid wall boundary condition and the analytical solution. Results will be shown for both sets of boundary conditions.

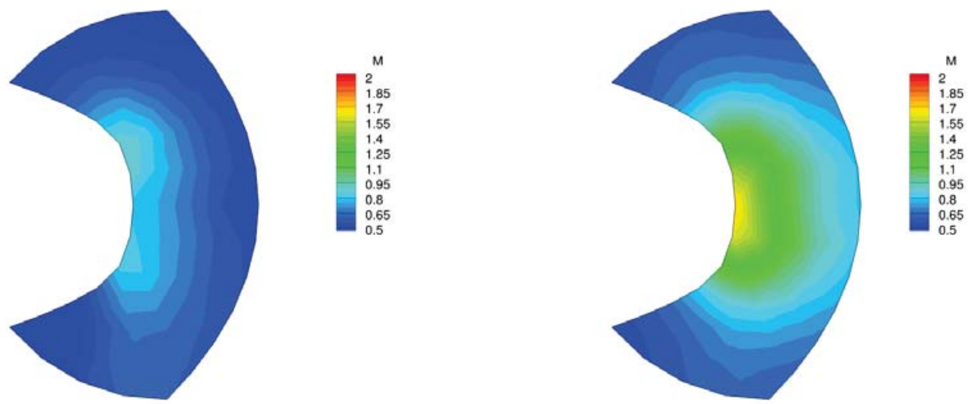
Table 1: Structured Domain Sizes

Grid Level	i Cells	j Cells
1	4	12
2	8	24
3	16	48
4	32	96
5	64	192
6	128	384

4 Results

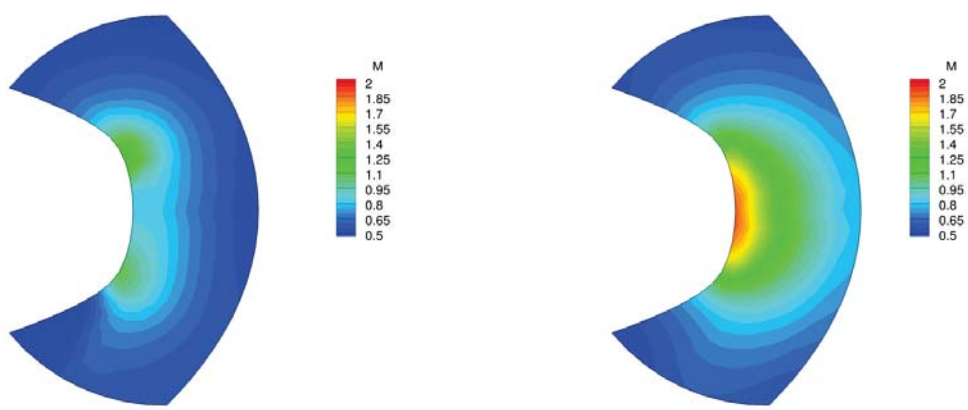
4.1 Contour Plots

Mach number contours of the converged solutions for each grid level are shown in Fig. 1 through 6. It can be seen that grid levels 1 and 2 are too coarse with the inviscid solid wall boundary condition at the walls, resulting in a converged solution with two inner-lobes rather than the expected one lobe. Also, it can be seen that using the analytical boundary condition at the walls improves the solution at all grid levels and eliminates the tendency to converge to a two inner-lobe solution, as shown in Fig. 1 and Fig. 2. Another thing noted by looking at the Mach number contours is the tendency for error to build up around the lower-half of the inner wall. This is due to a combination of the boundary condition and from representing the smooth-curved wall geometry by linear elements. Thus using a more exact boundary condition (via using the analytical solution) and/or using a finer approximation of the smooth-curved wall geometry (via more grid cells along the wall contour) reduced the buildup of error along the inner wall.



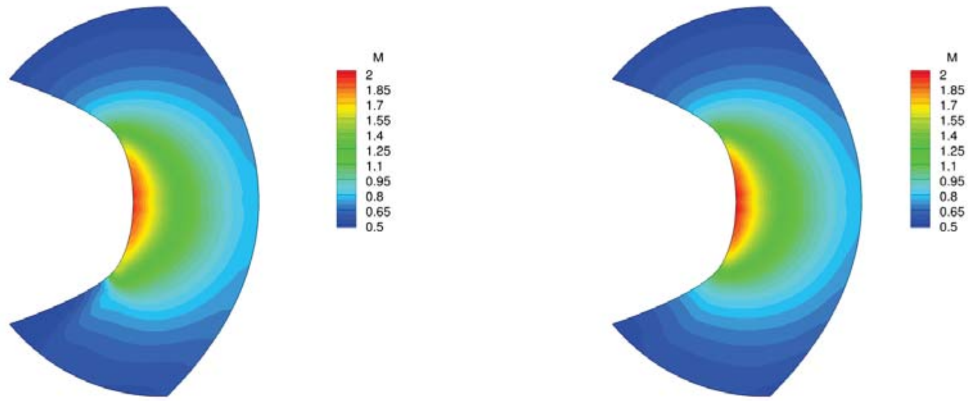
(a) Solid wall boundary condition. (b) Analytical boundary condition.

Figure 1: Mach number contours for the level 1 grid.



(a) Solid wall boundary condition. (b) Analytical boundary condition.

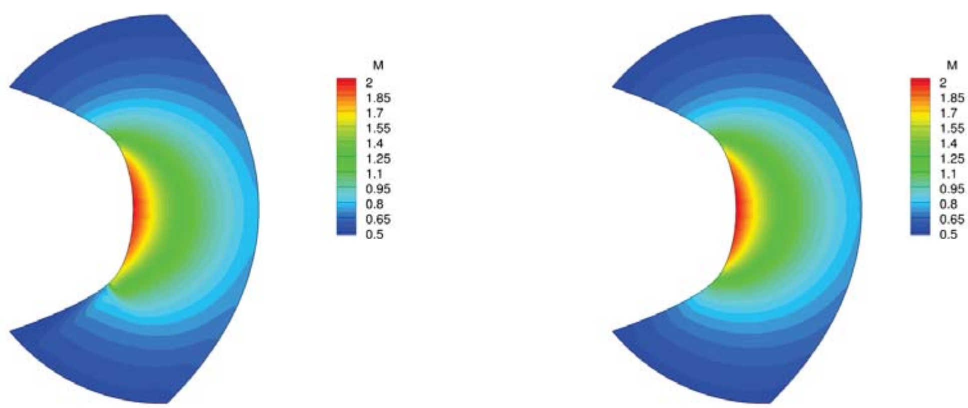
Figure 2: Mach number contours for the level 2 grid.



(a) Solid wall boundary condition.

(b) Analytical boundary condition.

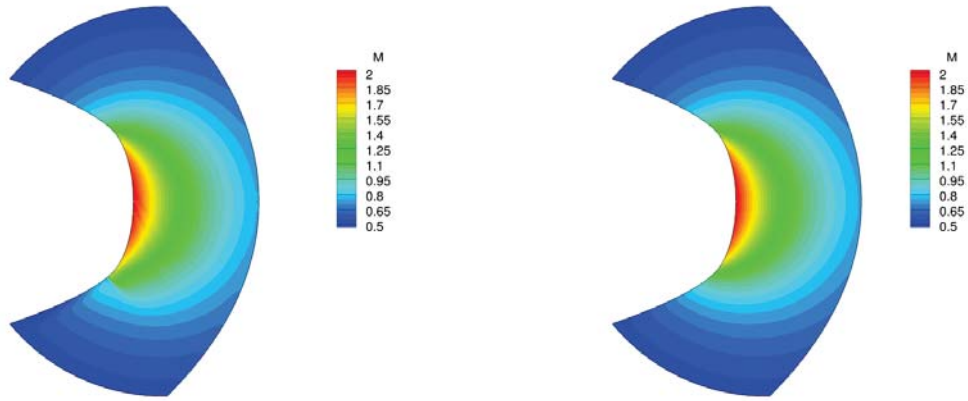
Figure 3: Mach number contours for the level 3 grid.



(a) Solid wall boundary condition.

(b) Analytical boundary condition.

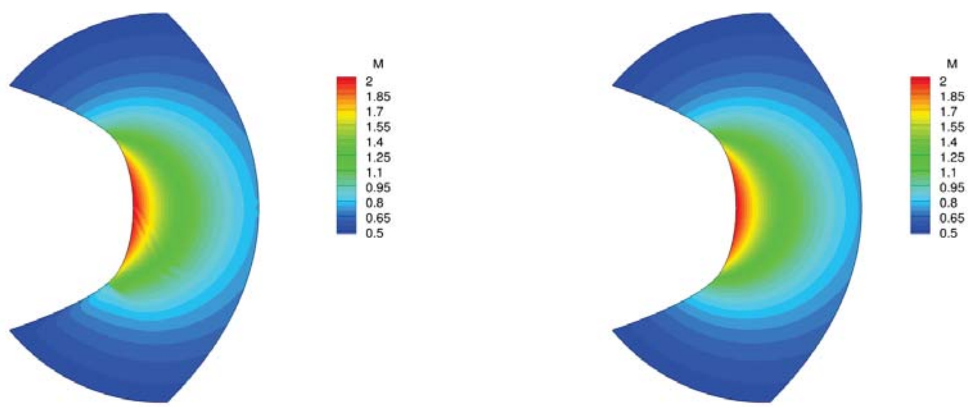
Figure 4: Mach number contours for the level 4 grid.



(a) Solid wall boundary condition.

(b) Analytical boundary condition.

Figure 5: Mach number contours for the level 5 grid.



(a) Solid wall boundary condition.

(b) Analytical boundary condition.

Figure 6: Mach number contours for the level 6 grid.

4.2 Workshop Metrics

The workshop requires the L2 norm of the entropy error of the converged solution to be computed and compared to the work units and length scale for each grid. The entropy L2 norm was computed as follows:

$$Error_{L2(s)} = \sqrt{\frac{\sum\limits_{i=1}^{N} \int\limits_{V_i} \left(s_i - s_{exact,i}\right)^2 dV}{\sum\limits_{i=1}^{N} |V_i|}} \tag{1}$$

$$s = \frac{p}{\rho^\gamma} \tag{2}$$

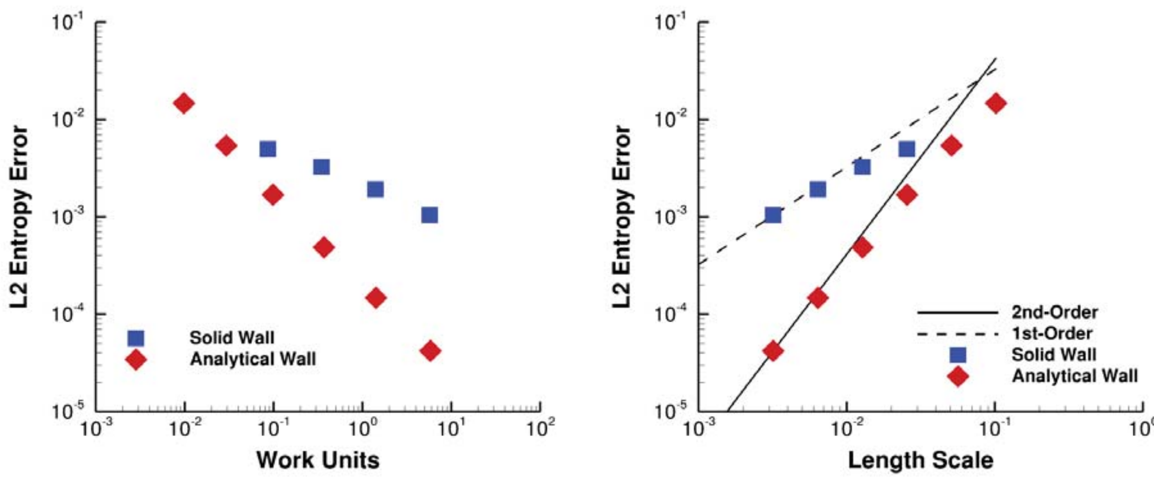
The work units were defined as the time it took ez4d to run 100 iterations normalized by the time it took to run the TauBench executable.

$$WorkUnit = \frac{t_{ez4d,100}}{t_{TauBench}} \tag{3}$$

Following suite, the length scale was defined as follows:

$$h = \frac{1}{\sqrt{nDOF}} = \frac{1}{\sqrt{n_{cells}}} \tag{4}$$

Figure 7 shows the entropy error verses work units and length scale. It can be seen that while the entropy error decreases with mesh refinement (represented by decreasing length scale and increasing work units) for both boundary conditions at the wall, the rate of decrease of the entropy error is different between the two boundary conditions. Also as expected, solutions using the analytical solution boundary condition have less entropy error than solutions using the inviscid solid wall boundary condition per given grid level.



(a) Entropy error verses work units.

(b) Entropy error verses length scale.

Figure 7: Workshop metric results.

Acknowledgments

The author would like to thank the Aerosciences Project under the NASA Fundamental Aeronautics Program for support with this effort as well as Chau-Lynn Chang and Balaji Venkatachari for guidance and support with using the ez4d code.

References

[1] Chang, S. C., "The Method of Space-Time Conservation Element and Solution Element - A New Approach for Solving the Navier-Stokes and Euler Equations," *Journal of Computational Physics*, Vol. 119, No. 2, 1995, pp. 295–325.

[2] Chang, S. C., Wang, X. Y., and Chow, C. Y., "The Space-Time Conservation Element and Solution Element Method - A New Resolution and Genuinely Multidimensional Paradigm for Solving Conservation Laws," *Journal of Computational Physics*, Vol. 156, No. 1, 1999, pp. 89–136.

[3] Chang, C. L., "Time-Accurate, Unstructured-Mesh Navier-Stokes Computations with the Space-Time CESE Method," Tech. Rep. AIAA 2006-4780, July 2006.

[4] Venkatachari, B., Cheng, G., and Chang, S. C., "Development of a High-Order CESE Scheme For Transient Viscous Flows," Tech. Rep. AIAA 2009-3984, June 2009.

[5] Chang, S. C., "A New Approach for Constructing Highly Stable High Order CESE Schemes," Tech. Rep. AIAA 2010-543, January 2010.

[6] Chang, C. L., Venkatachari, B., and Cheng, G., "Time-Accurate Local Time Stepping and High-Order Space-Time CESE Methods for Multi-Dimensional Flows with Unstructured Meshes," Tech. Rep. AIAA 2013-3069, June 2013.