Review of: "Aerodynamic Design and Performance Analysis of Mars Ascent Vehicles"

Janusz Piechna[1]

1 Technical University of Warsaw

Potential competing interests: No potential competing interests to declare.

The authors describe a numerical scheme for solving the Euler equations using the finite difference method supplemented with viscosity and turbulence models.

The authors use turbulence models that are described by transport equations, at the same time without providing any values used by these equations at the inlet to the computational area.

Text weakness:

The authors did not test the independence of the numerical solution from the grid density.

Nothing is known about the y+ value.

The use of a laminar flow model and the use of a turbulent flow model do not mean that in reality such flows in the studied object flow structure occur. In general, each of the fluid flow models should give some indication of a change in flow structure. Turbulence models are described by transport equations. Thus, inlet conditions affect the development of turbulence on the streamlined surfaces. I have the impression that the authors do not remember this.

As the analysis goes on to show, a different turbulence model is required to obtain calculation results that are consistent with experimental test results.

Why, then, do they pay so much attention to the presentation of non-physical results?

Careless text editing. The tables contain numerical values broken into two lines, which causes confusion for the reader and excites them when they realize that the Mach number is 6.02 and not 6.0.

No links to the literature list.

The paeans to Tecplot are interesting but irrelevant.

Figures 10-14 show Mach number distributions in the plane of symmetry mainly in the area above the object, with no detail in the area below the object, which continues to be an area of particular interest.

Figures 14-24 show dramatically different results for the laminar and turbulent flow models.

What conclusions can be drawn from this? Which model reflects reality? What are the implications of this?

Only by comparing the experimental data with a turbulence model that takes into account the laminar-turbulent transition process does it provide any picture close to the reality presented by the experimental results.

The authors deliberate on the influence of vortex structures. There are methods to visualize vortices: Q-parameter, lambda 2, omega. Instead of conjecture, one can show them. And draw conclusions.

It seems that the authors have put too much effort into a broad description of the mathematical model, which in some parts is interesting, and in others shows elements that are commonly known.

The authors' focus on thermal problems is important. On the other hand, the lack of information on the boundary conditions on the vehicle walls somewhat weakens this relevance. It can be presumed that this is an isothermal condition.

They show in Figures 14-24 information that is incomparable.

It seems that the discrepancies in the CFD and experimental results at the end of the vehicle are due to incompletely identical boundary conditions at the outlet. The pressure information behind the vehicle moves upstream through the wall layer. The authors did not state what pressures are behind the vehicle in the experimental tests and CFD calculations.

Considerations and conclusions.

"The fluid temperature remains constant across the bow shockwave," the authors wrote. Not quite, only the stagnation temperature remains constant.

Conclusions very general and not based on presented results.

The authors refer to the changing Reynolds number. OK. They give the value of Re in the free area in units of 1/m. OK. How, then, do they determine the Reynolds number in the environment of the vehicle?

They use very complicated software, showing only elementary results. They did not use the full potential of the solver at their disposal.

I would suggest adding a conclusion - Laminarize the flow to reduce heat generation. In incompressible flow, there are several ways to laminarize - changing the shape of the surface, grooving the surface. Let's try to find something similar in compressible flow.

I have doubts about the layout of the paper. The authors split the results into a group of laminar flow and assumed turbulent flow and a second group of solutions with a laminar-turbulent transition model.

Such a division undermines the sense of the calculations in the first group. It seems that all used flow models should be compared simultaneously.

I understand that there is no full set of results. OK. But?

The paper seems to be interesting enough to publish, but I would suggest making modifications after considering the comments posted above.