Transonic and Supersonic Flows Modeling in ANSYS CFX-5.7 Program

Natalia Vladimirova ,Michael Starodubcev

Gas dynamic problem description:

Numerical modeling of outer transonic and supersonic flows around the missile, the CAD model of the missile obtained from CATIA V5 is shown in Fig.1.

Numerical computations were carried out on personal computer in ANSYS CFX-5.7 program, computational meshes were generated in ANSYS ICEM CFD program.

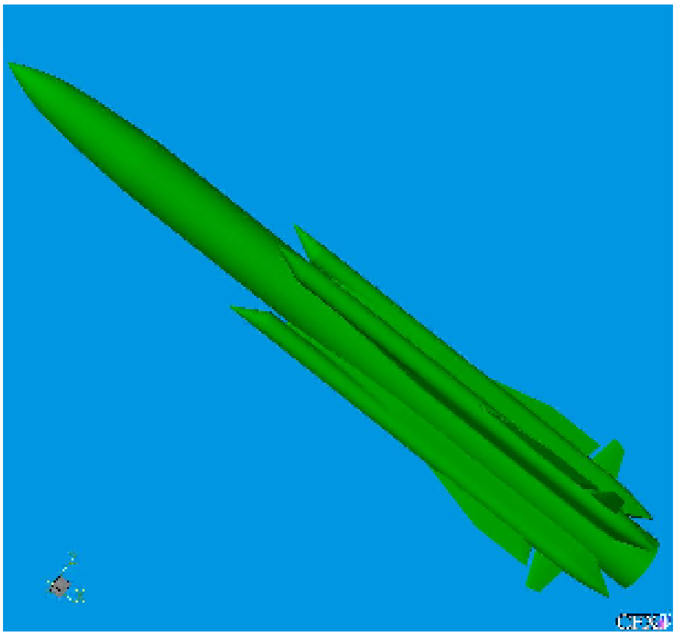

Fig. 1. CAD model of the missile

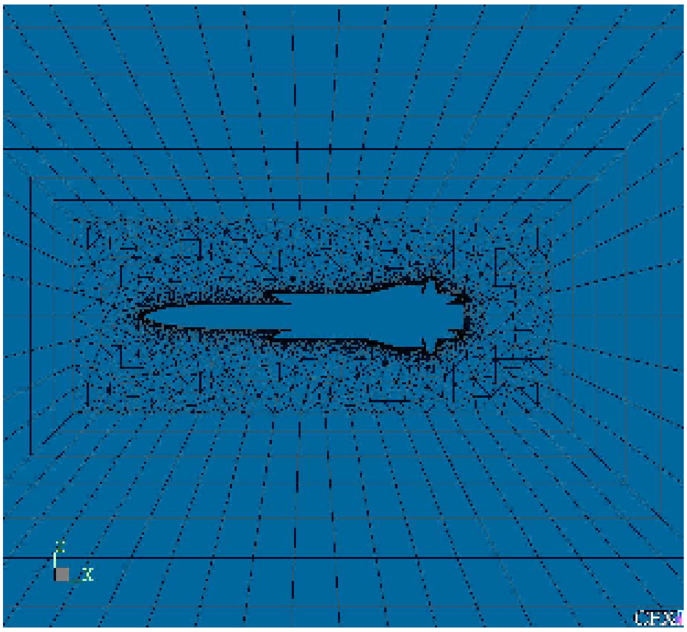

Fig. 2. Hybrid unstructured mesh in the symmetry plane

The three-dimensional Navier-Stokes equations described compressible viscous turbulent flows were numerically solved by ANSYS CFX-5.7 program.

The finite volume method, node centered high-resolution scheme for convective and viscous parts and *SST k-ω* turbulence model for separated flows modeling were used. Computational domain was the parallelepiped with the facets distant from missile surface up to 8 calibers.

On the facets of the parallelepiped simulating infinite computational domain full of air flow there were used following boundary conditions: “Inlet”, “Outlet” (for supersonic regimes), “Opening” (for subsonic regimes), “Symmetry” and ”Wall” (with “slip” on the side facets).

Three-dimensional unstructured hybrid mesh was created in *Tetra* and *Hexa* modules of ANSYS ICEM CFD program (Fig. 2).

Generated mesh consisted of 107 607 nodes, 494 768 cells (485 248 tetrahedrons, 9 044 hexahedrons, 476 pyramids), at the missile surface triangular mesh was generating. In the boundary layers near the body surface and at the regions of large curvature variations (sharp nozzles and sharp wing edges, Fig. 3-4) the mesh cells were refined and pressed down.

The choice of such mesh topology is conditioned by complex missile geometry. Generating multi-block structured hexahedron mesh for such complex geometrical configurations is awful problematic and ponderous.

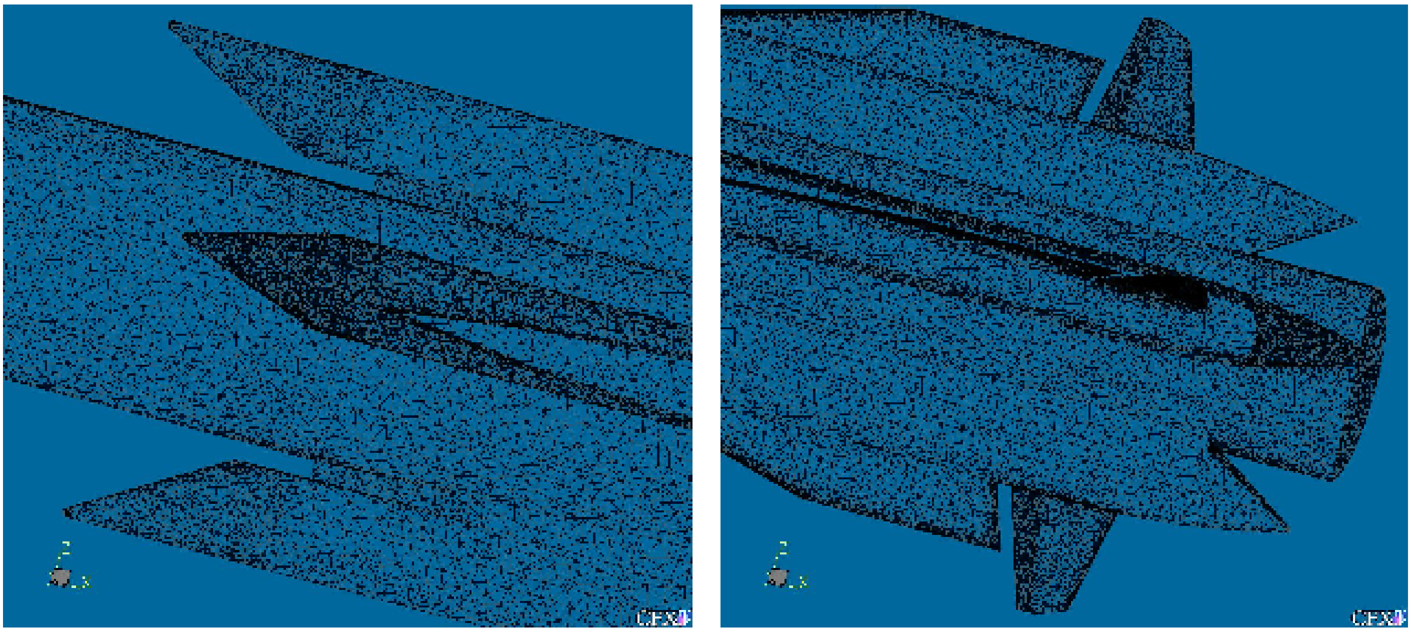

Fig. 3. Triangular mesh fragments on the missile surface (wings, fins, pylons, spinners)

Numerical calculations of integral and distributed missile aerodynamic parameters were realized for angle of attack $\alpha=6°$, Mach number M=0.8; 1.2; 3.0 in the Reynolds number $Re=(24\text{-}90)\cdot 10^6$ diapason.

Numerical results and correlations with experimental date are depicted in Fig. 4-9.

As an analysis of numerical data shows even for sufficient rare unstructured hybrid mesh (about 100 000 nodes) generated for complex geometry it works out to get sufficient positive numerical integral results correlated with experiment adequately.

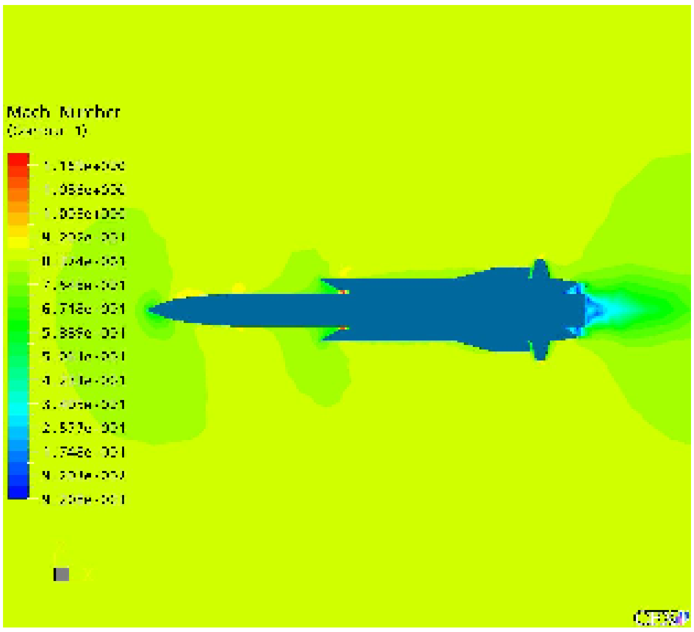

Fig. 4. Mach number distribution at the symmetry plane, M=0.8, α=6°

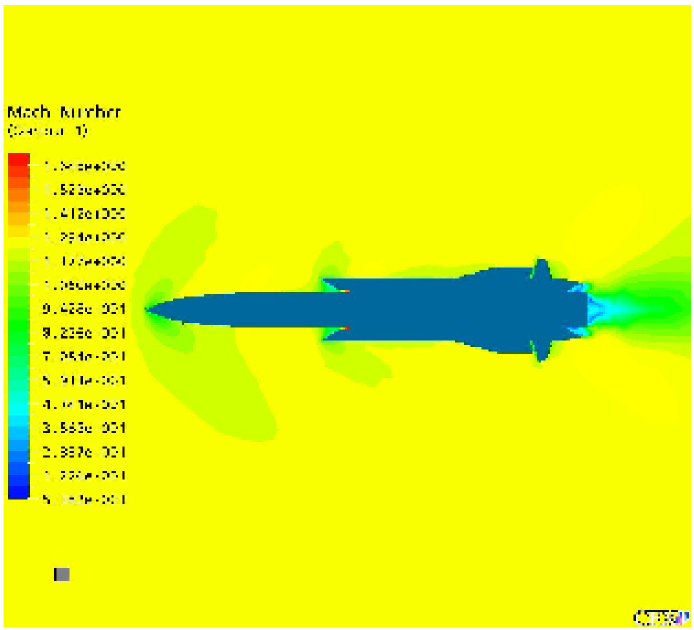

Fig. 5. Mach number distribution at the symmetry plane, M=1.2, α=6°

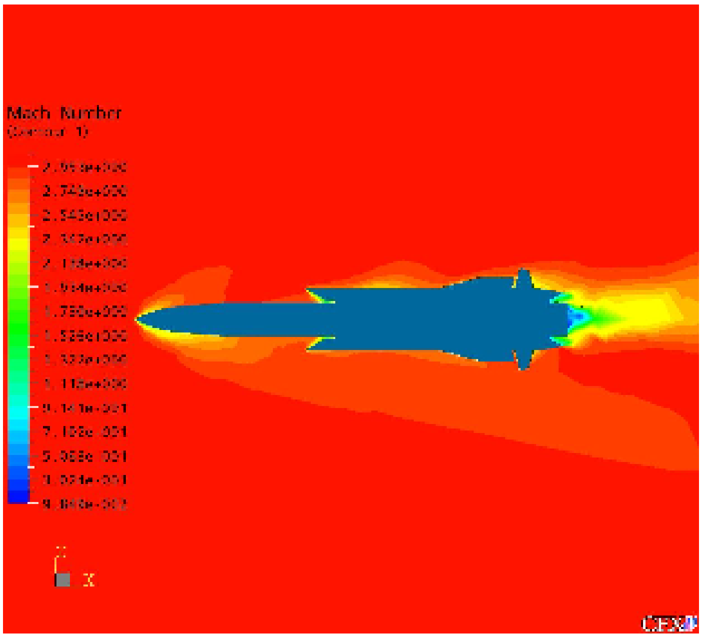

Fig. 6. Mach number distribution at the symmetry plane, M=3.0, α=6°

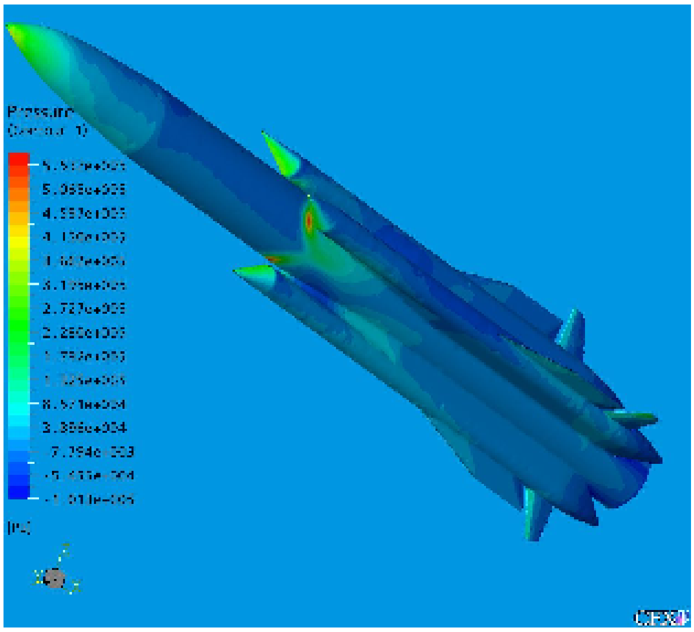

Fig. 7. Pressure distribution at the missile surface M=3.0, α=6°

As we can see from Fig. 8-9 the character of dependence on Mach number calculated in CFX-5.7 lift coefficient C_l and pitching moment C_m is consistent with experimental relations well. Somewhat overvalued calculated values of C_l in comparison with their experimental values are possibly conditioned insufficient mesh size and consequently oversized cells in the separated regions and insufficient appressed mesh against body walls in the boundary layers.

To achieve results that are more accurate and better correlations between numerical and experimental data it needs to actualize the computations on the computer cluster with 8-10 Gbytes of operative memory using computational meshes with 2-5 millions of nodes.

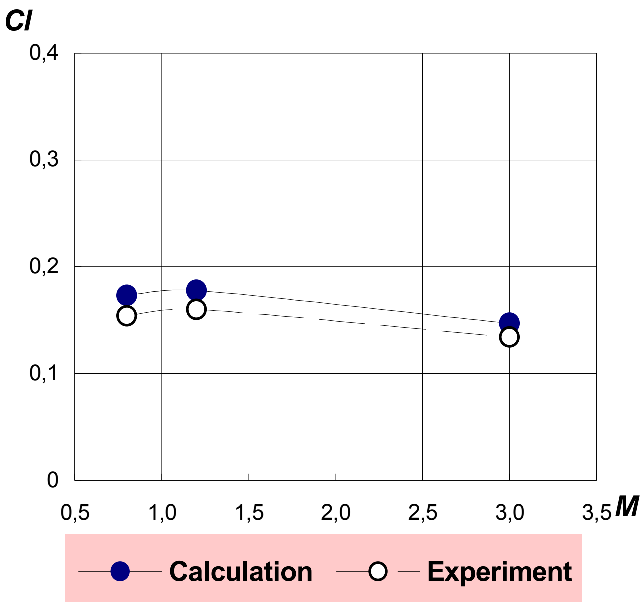

Fig. 8. The correlation between lift coefficient and Mach number (calculation and experiment)

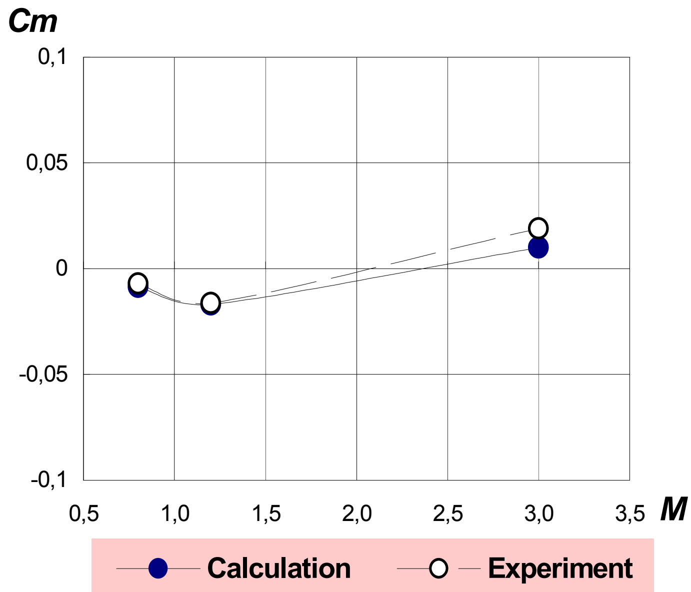

Fig. 9. The correlation between pitching moment coefficient and Mach number (calculation and experiment)