Test Calculation Results of ANSYS CFX 5.7.1 Simulation of Transonic and Supersonic Flow around Vehicle

Natalia Vladimirova

Problem description

The vehicle model consists of axisymmetric cylindrical hull with advance ogive part, sharp nose, end part with flat base section and horizontal rectangular fins.

The fins have typical for supersonic configurations diamond sections with cuneal leading and trailing edges.

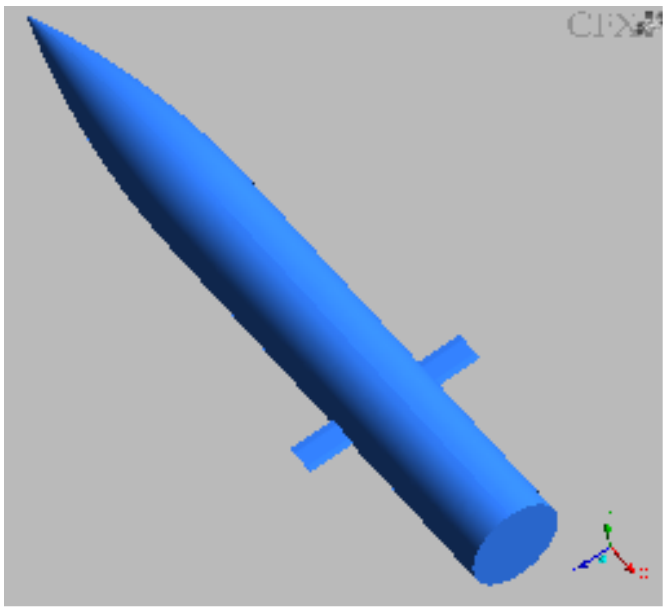
The external viscous flow around vehicle model is calculated in ANSYS CFX 5.7.1 program for Mach number M=0.8, 1.0, 1.7, Reynolds number Re=(14-30)·10^6 diapason, angles of attack α=6°, 16° and angles of fin deflection δ=0 и 21°.

The computational domain is parallelepiped, the size of the domain is 10 calibres in all directions, the ideal gas at the normal conditions (T=300°, P=1 atm) was used as work environment.

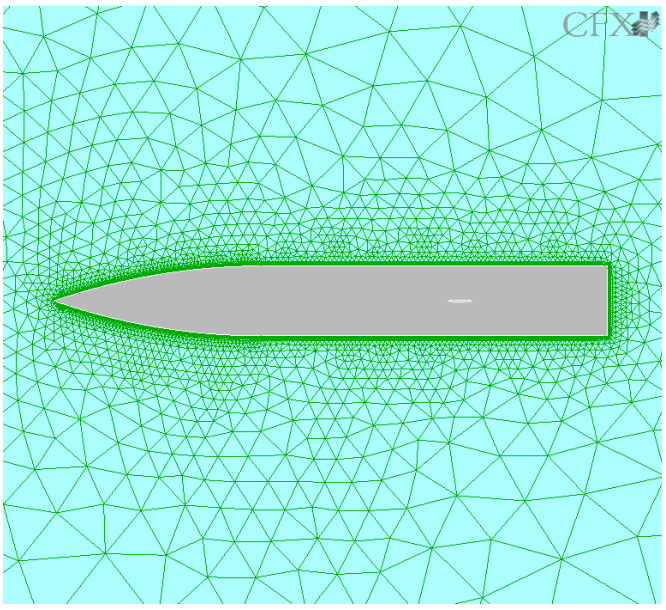
Two unstructured hybrid tetra/wedges meshes (for two fin deflection configurations appropriately) were generated in the ANSYS CFX-Mesh module.

Generated mesh consisted of 425000 nodes on an average. In the boundary layers near the body surface and at the vicinity of surface fractures provocative viscous separation the mesh cells were refined and pressed down.

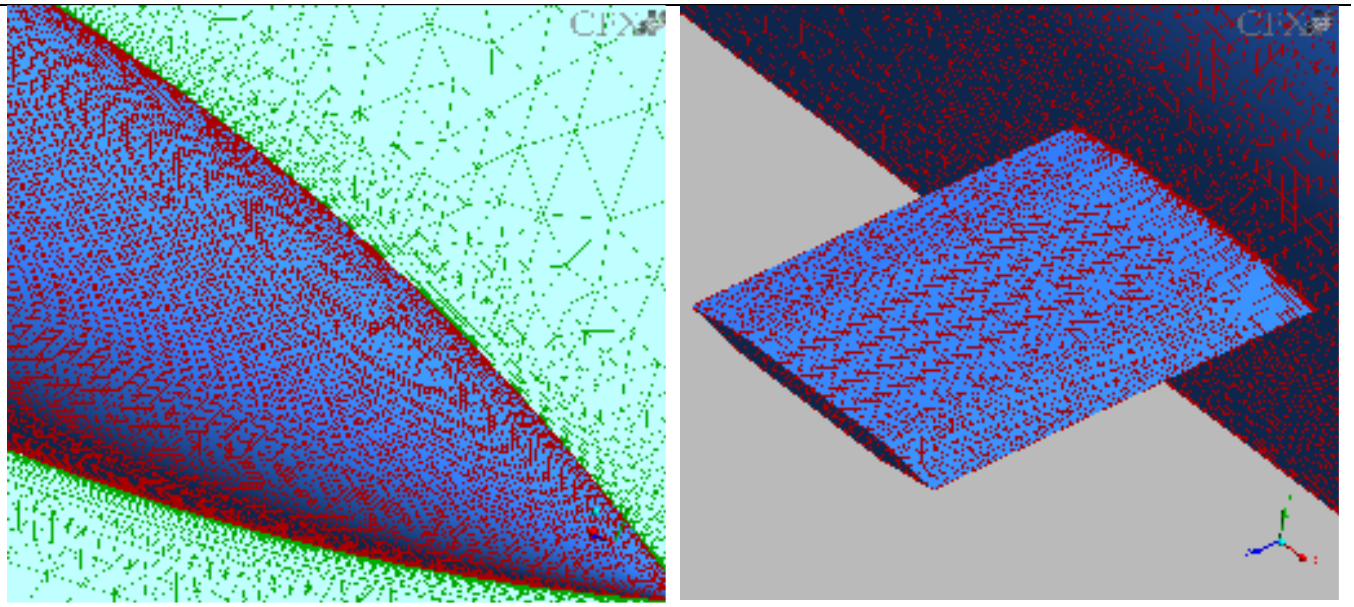
CAD-model of vehicle and computational mesh fragments in the symmetry plane, on the nose surface and fin surface are depicted in Figure 1.



CAD-model



Computational mesh in the symmetry plane



Computational mesh near the body nose *Surface mesh at the horizontal fin*

Figure 1. CAD-model and computational mesh fragments

Numerical calculations were carried out in ANSYS CFX-5.7.1 program on Intel Pentium 4 CPU 2.00 GHz personal computer with 1 GB of operative memory. For solving three-dimensional Navier-Stokes equations describing compressible viscous turbulent flows the finite volume method, node centered high-resolution scheme for convective and viscous parts and *SST (Shear-Stress-Transport) k-ω* turbulence model for separated flows modeling were used.

To obtain steady-state convergent solution for one flow regime (Mach number and angle of attack are preassigned) 50 –100 time iterations are usually required, this corresponds 10 – 20 hours of brainwork of Pentium 4 class PC for computational meshes with about 500 000 nodes.

As a result of computational flow simulation integral (axial and normal pressure and viscous forces) and distributed (pressure coefficient, Mach number field) aerodynamic characteristics are defined for vehicle in toto and its elements (fins, base, nose).

Figures 2, 3 show calculation and experimental (from aerodynamic wind tunnel tests) results correlations for fin normal force coefficient Cn.

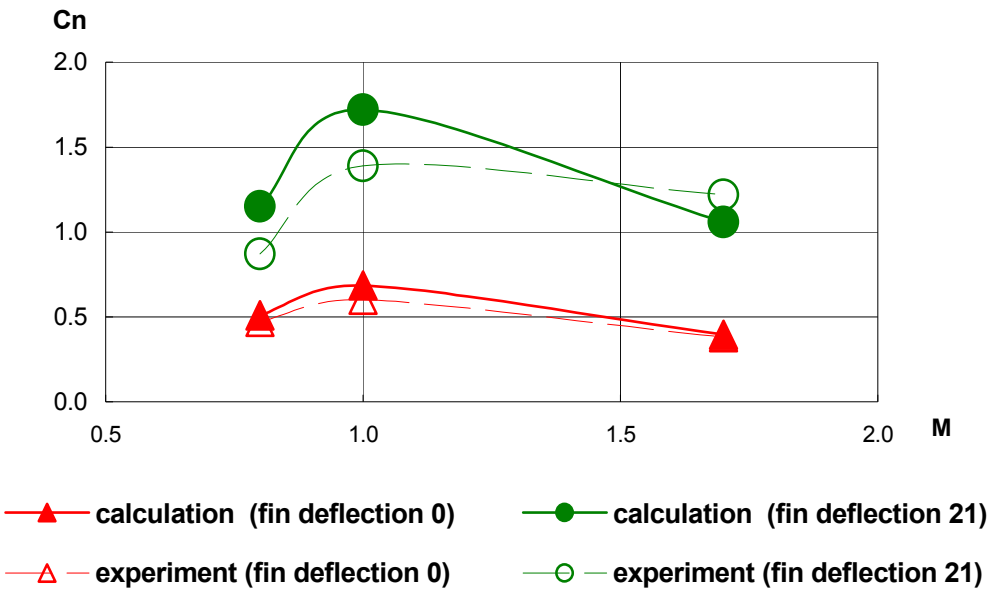


Figure 2. Fin normal force coefficient Cn, calculation and experiment (angle of attack α=6°)

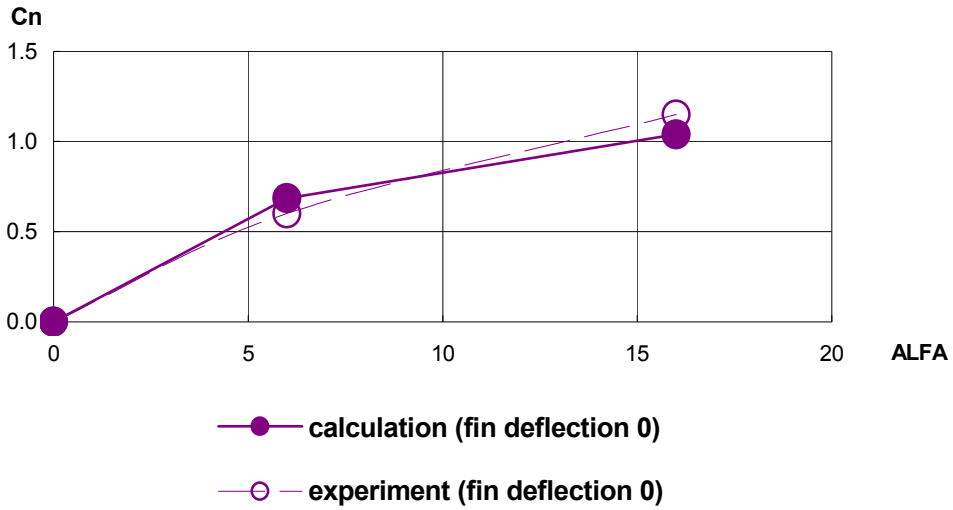
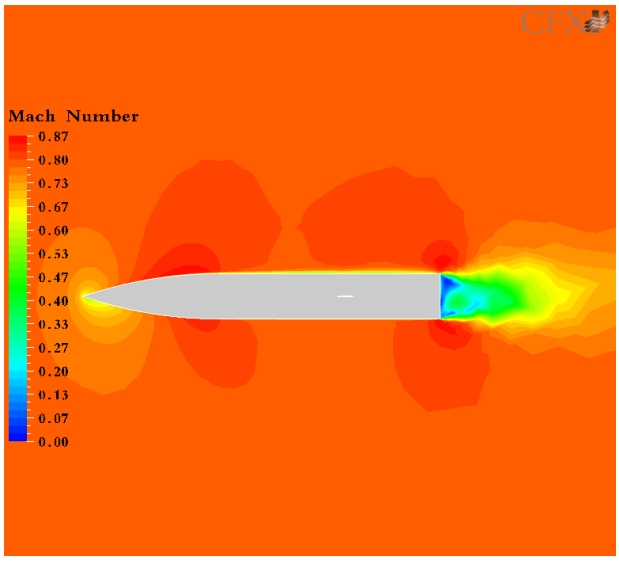
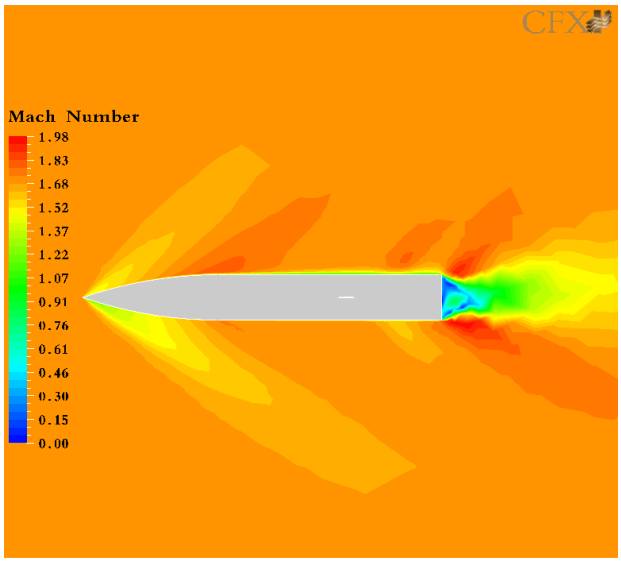


Figure 3. Fin normal force coefficient Cn, calculation and experiment (M=1, fin deflection δ=0°)

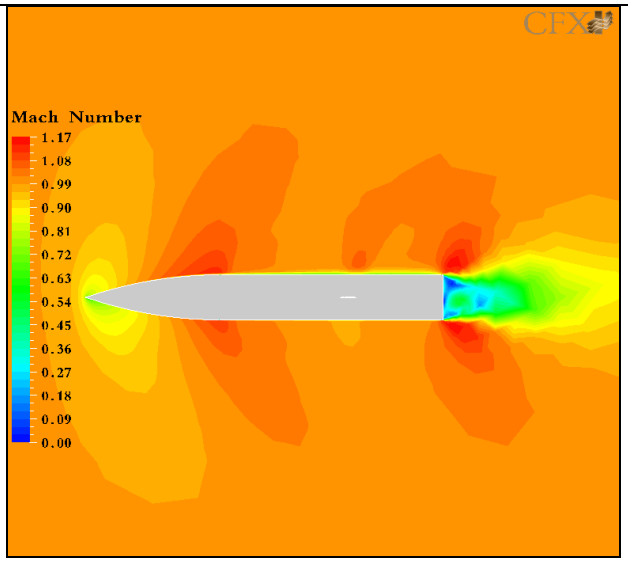
Figures 4, 5 show calculation results for Mach number distribution at the symmetry plane of computational domain and pressure coefficient profiles and Mach number distribution at the board and tip fin sections.



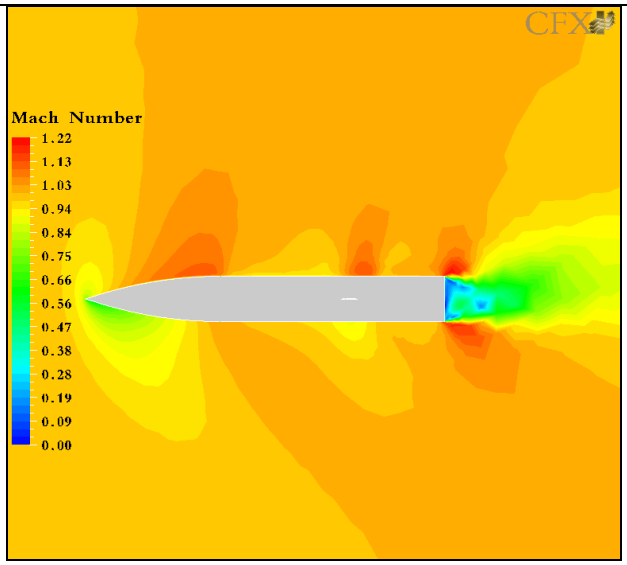
M=0.8 α=6°



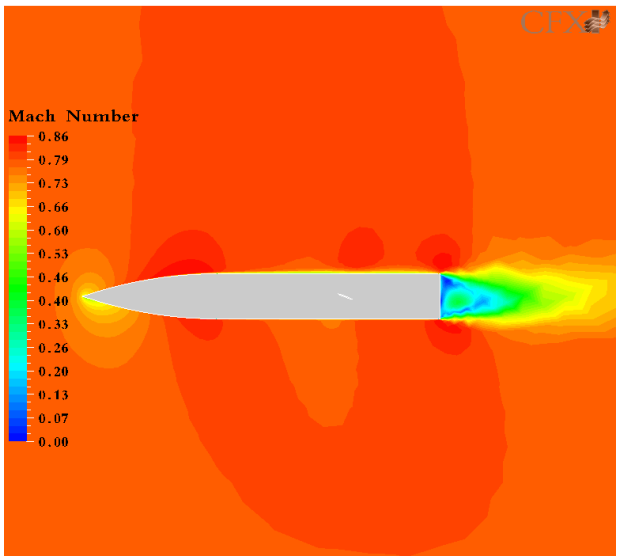
M=1.7 α=6°



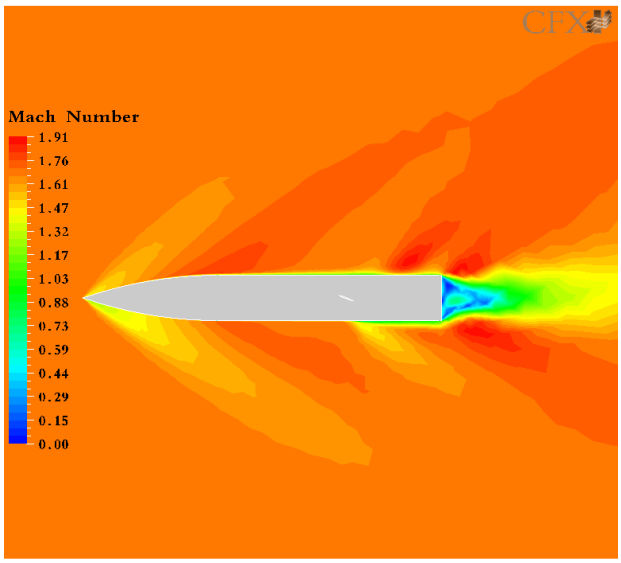
M=1.0 α=6°



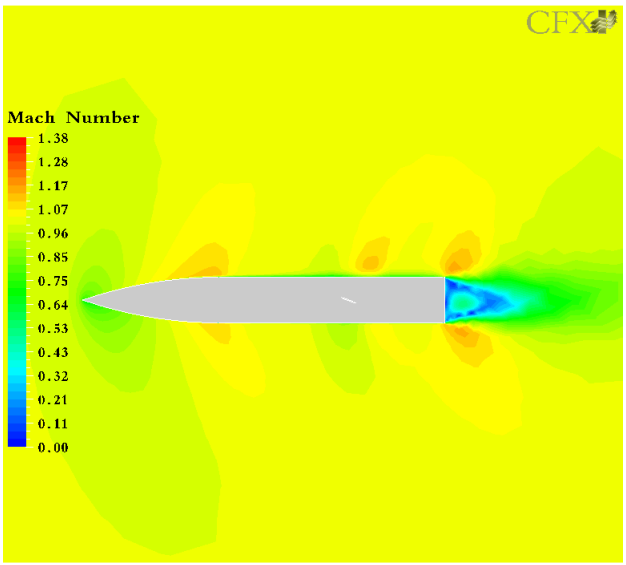
M=1.0 α=16°



M=0.8 α=6°



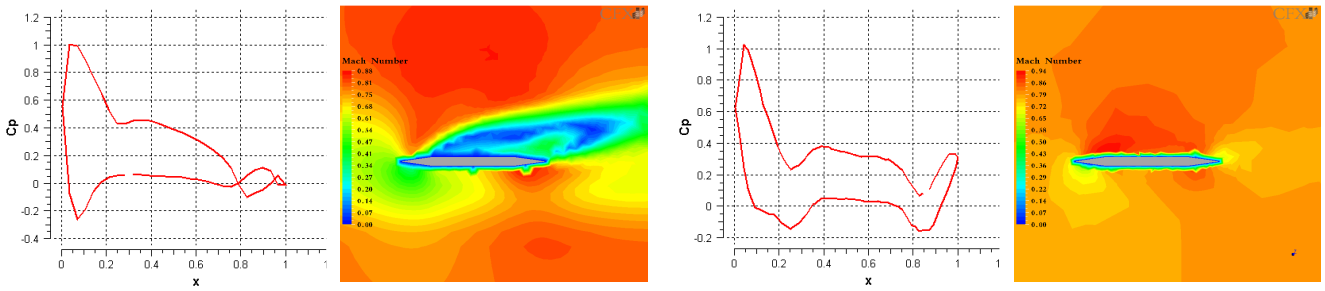
M=1.7 α=6°



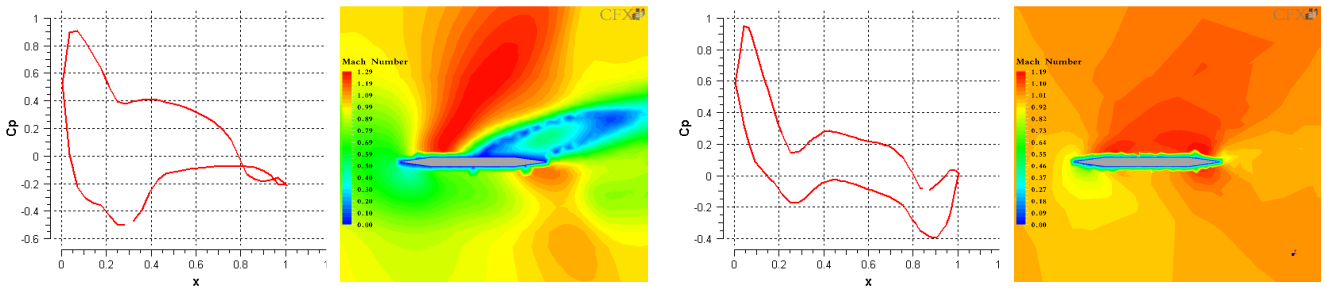
M=1.0 α=6°

Figure 4. Mach number distribution at the symmetry plane (fin deflection δ=21°)

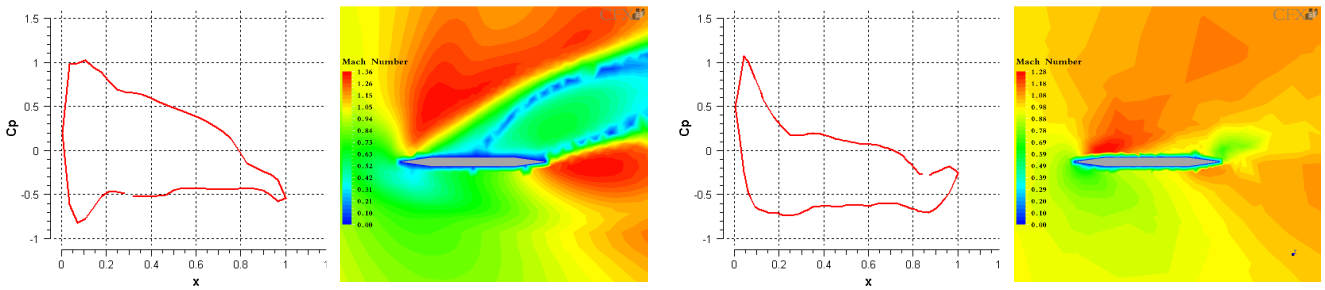
Board fin section *Tip fin section*



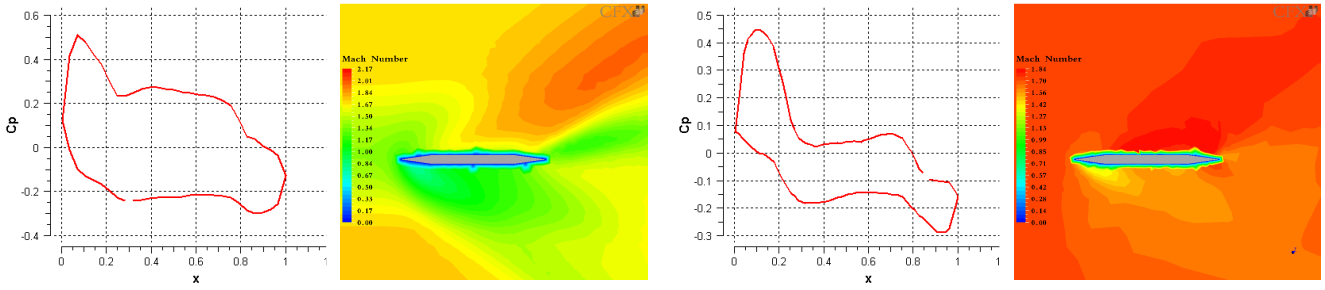
M=0.8 α=6° δ=0°



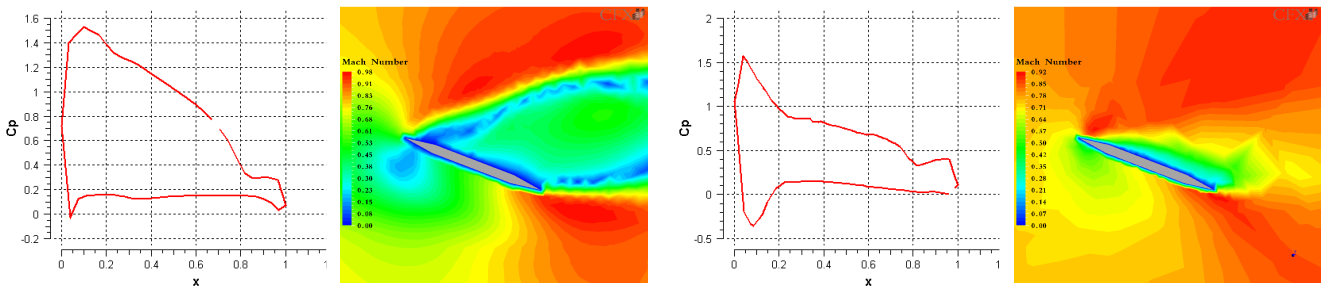
M=1.0 α=6° δ=0°



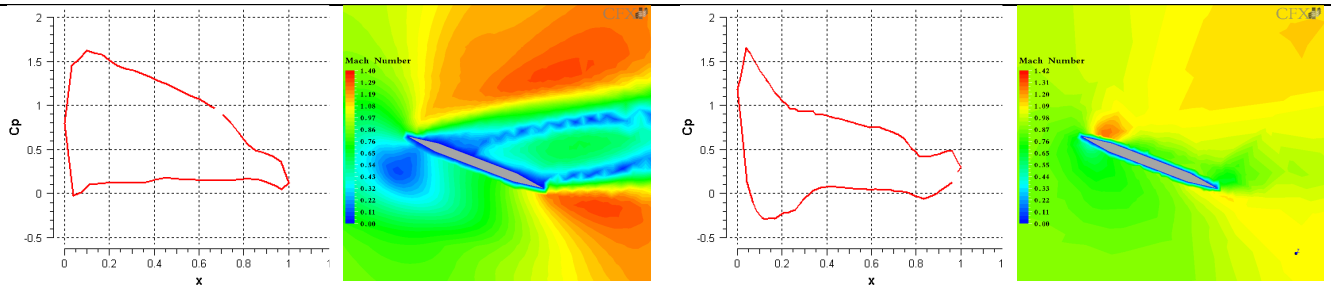
M=1.0 α=16° δ=0°



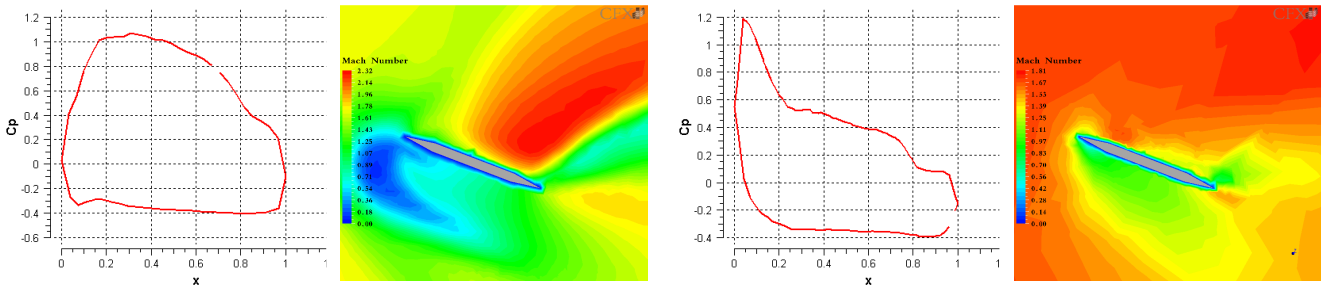
M=1.7 α=6° δ=0°



M=0.8 α=6° δ=21°



M=1.0 α=6° δ=21°



M=1.7 α=6° δ=21°

Figure 5. Pressure coefficient profiles and Mach number distribution at the board and tip fin sections

Analysis of results and conclusions

Analysis of ANSYS CFX 5.7.1 computational results and available experimental data denotes that considered problem of transonic, sonic and supersonic external flow around vehicle with horizontal fins is characterized complex flow features conditioned by gas compressibility and viscosity, sufficient powerful interference between the hall and the fins, separated flow about fins and flat base.

Even under the angle of attack α=6° on the upper surface of no deflected (δ=0°) fins the flow is fractionally separated, but correlation between computer results and experimental data is quite good.

Under the angle of fin deflection δ=21° the fin flow is globally detachable as for transonic and supersonic velocities and calculation – experiment correlation is down.

Personal computer test calculations show wide possibilities of ANSYS CFX 5.7.1 program in the context of aircraft and spacecraft aerodynamic calculations in the large diapason of free stream Mach numbers and angles of attack.

Analysis of computational and experimental data results in conclusion: to achieve more accurate computational results better correlating with 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.