Supersonic Detachable Flows Modeling in ANSYS CFX-5.7.1 Program: flow around bullet flying with the speed 800 m/s

Natalia Vladimirova ,Michael Starodubcev

Problem description

The problem of numerical simulation of supersonic viscous detached flow around the axisymmetric body with flat end (bullet or shell) is considered.

The bullet CAD model is shown in Figure1.

The bullet surface consists of cylindrical central part, advance ogive with blunt nose and oblique end part with flat base section.

The bullet length is 26 mm, the calibre is 8 mm.

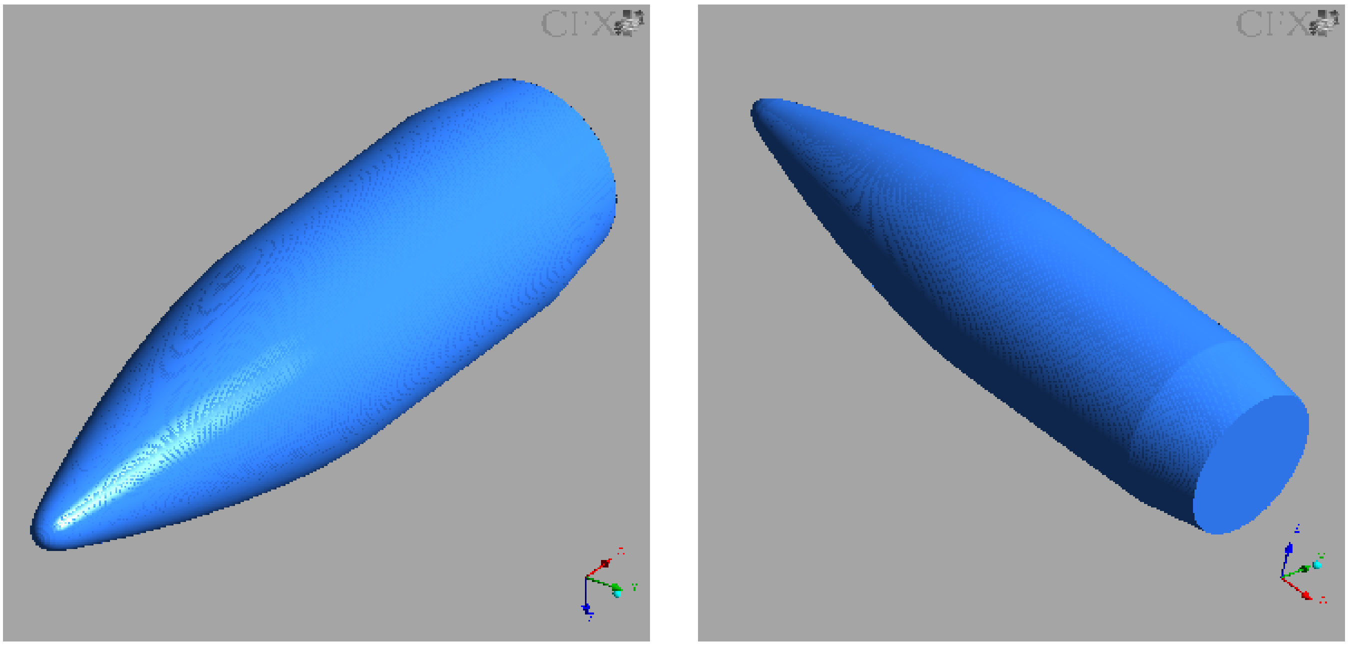

Figure 1. CAD model of the bullet

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; computational meshes were generated in ANSYS ICEM CFD program.

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 200 –250 time iterations are usually required, this corresponds 3.5 –4.5 ours of brainwork of Pentium 4 class PC (for computational meshes with about 100 000 nodes).

The bullet moves at zero angle of attack, so the problem is axisymmetric.

The computational domain is cylindrical segment, surrounding the bullet and full of air, the length of the domain is 14 calibres and segment angle is 10 degrees (Figure 2).

On the domain boundaries following boundary conditions were used: “Inlet”, “Outlet” (for supersonic regimes), ”Wall” (at the body surface and symmetry axis) and “Interface” (periodic conditions for conservation of fluxes across side faces of computational domain).

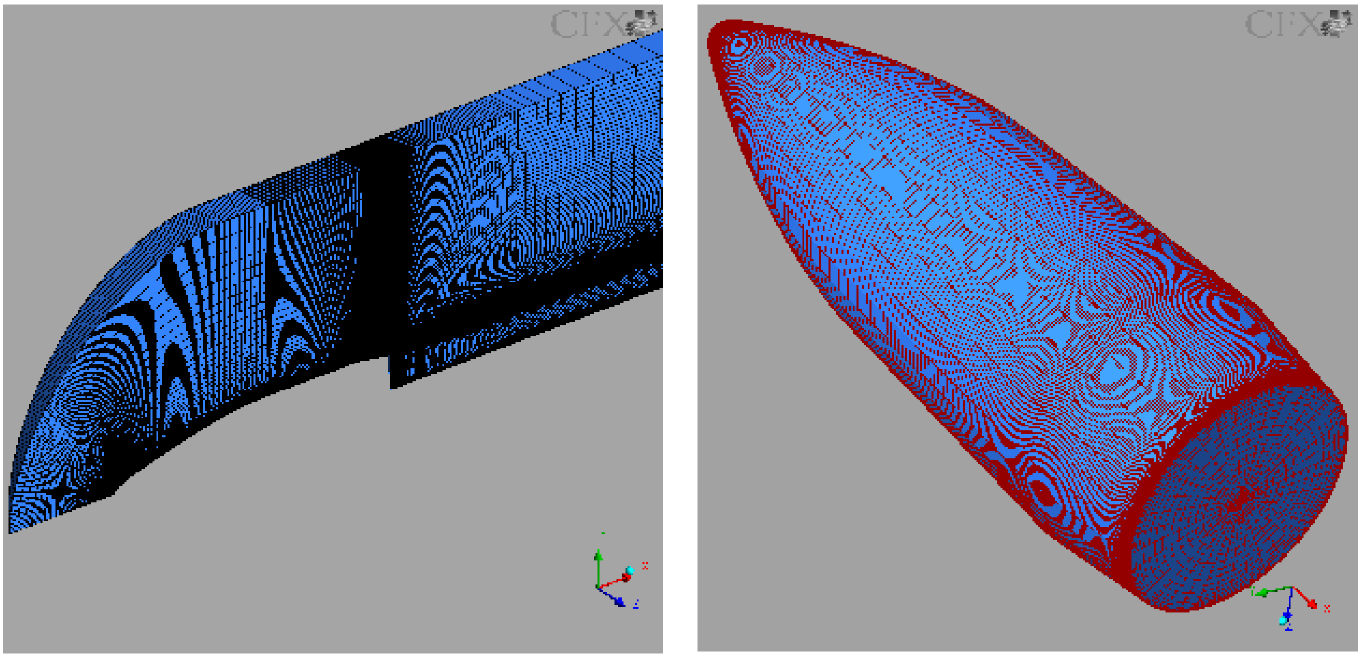

Figure 2. Computational domain and mesh (left), surface grid on the bullet (right)

The structural quad mesh in vertical plane of computational domain was generated in the ANSYS ICEM CFD 5.1 program (*Hexa* module).

Generated mesh consisted of 41471 nodes and 40953 quad cells.

Then this two-dimensional grid was "forced in" 10 degrees and "one-slice" three-dimensional hexahedral mesh with 82942 nodes was generated in the cylindrical coordinates.

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 (Figure 3).

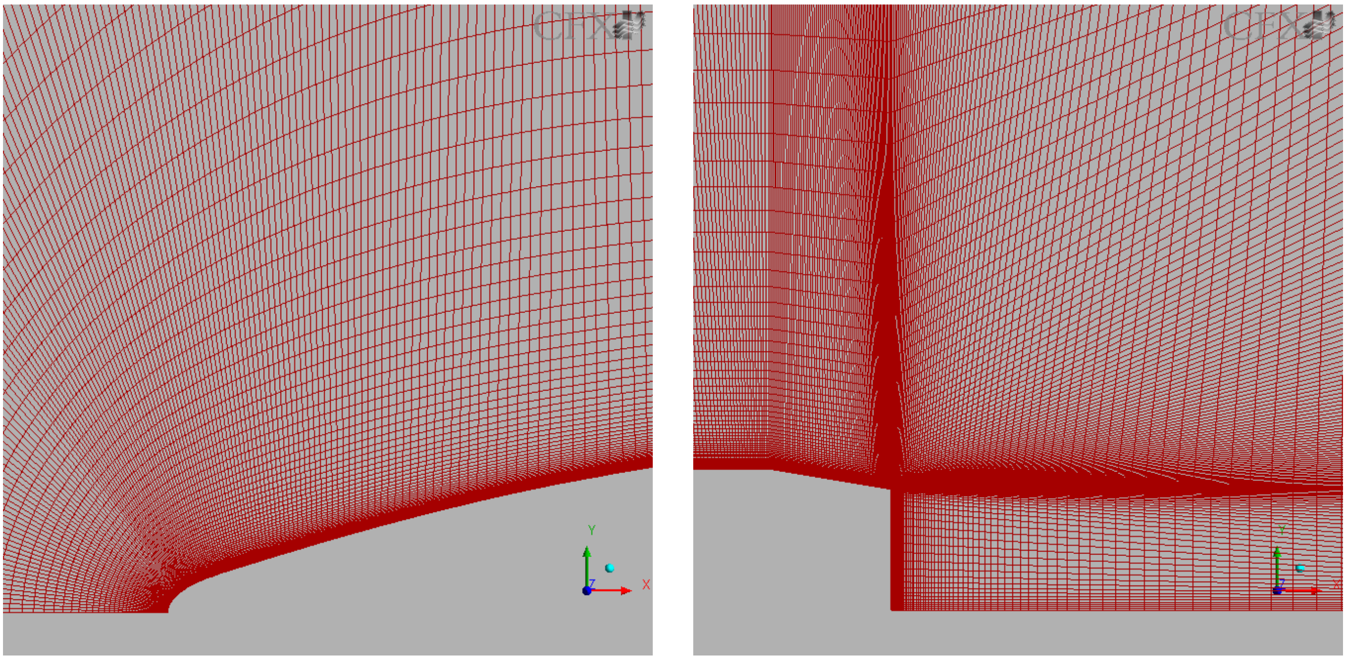

Figure 3. Fragments of refined and appressed mesh in the advance part (left) and in the base part (right)

Results

Numerical calculations of flow characteristics and bullet aerodynamic parameters were realized for angle of attack α=0°, Mach number M=2.38 and Reynolds number Re=$1.4 \cdot 10^6$.
Numerical results and some experimental data are depicted in Figures 4-7.

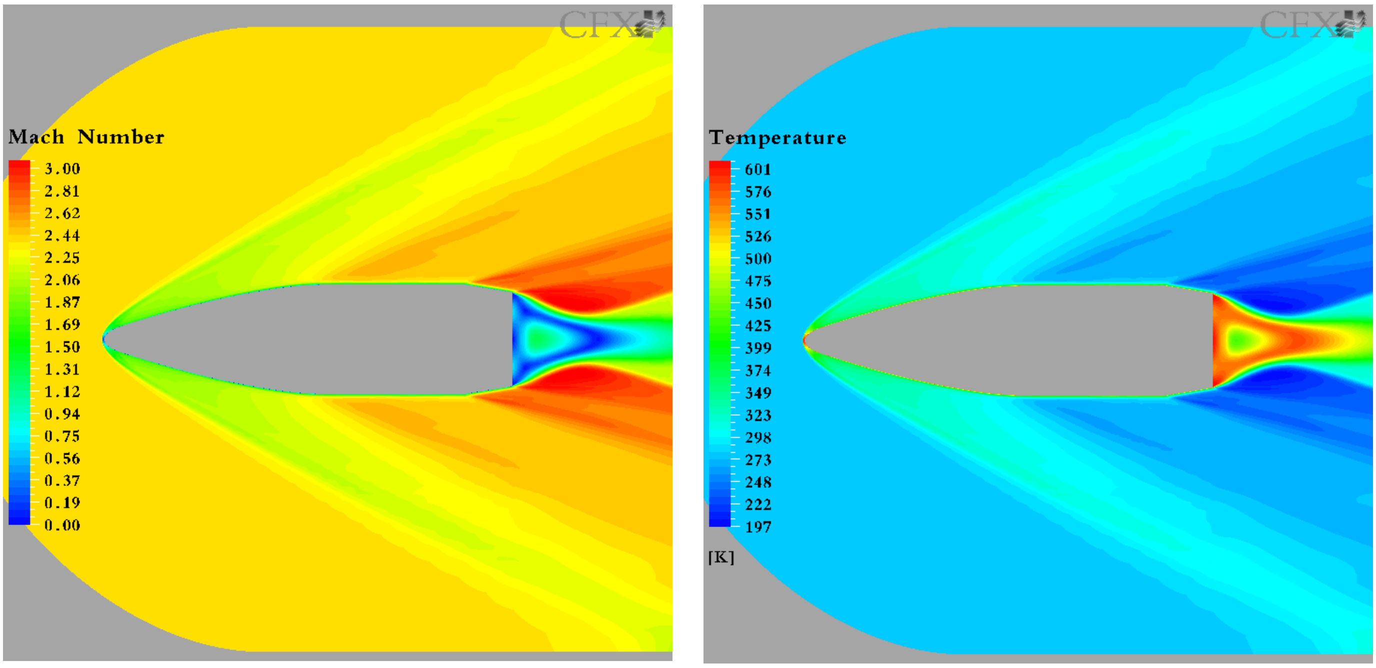

Figure 4. Mach number distribution (left) and temperature distribution (right) in the symmetry plane

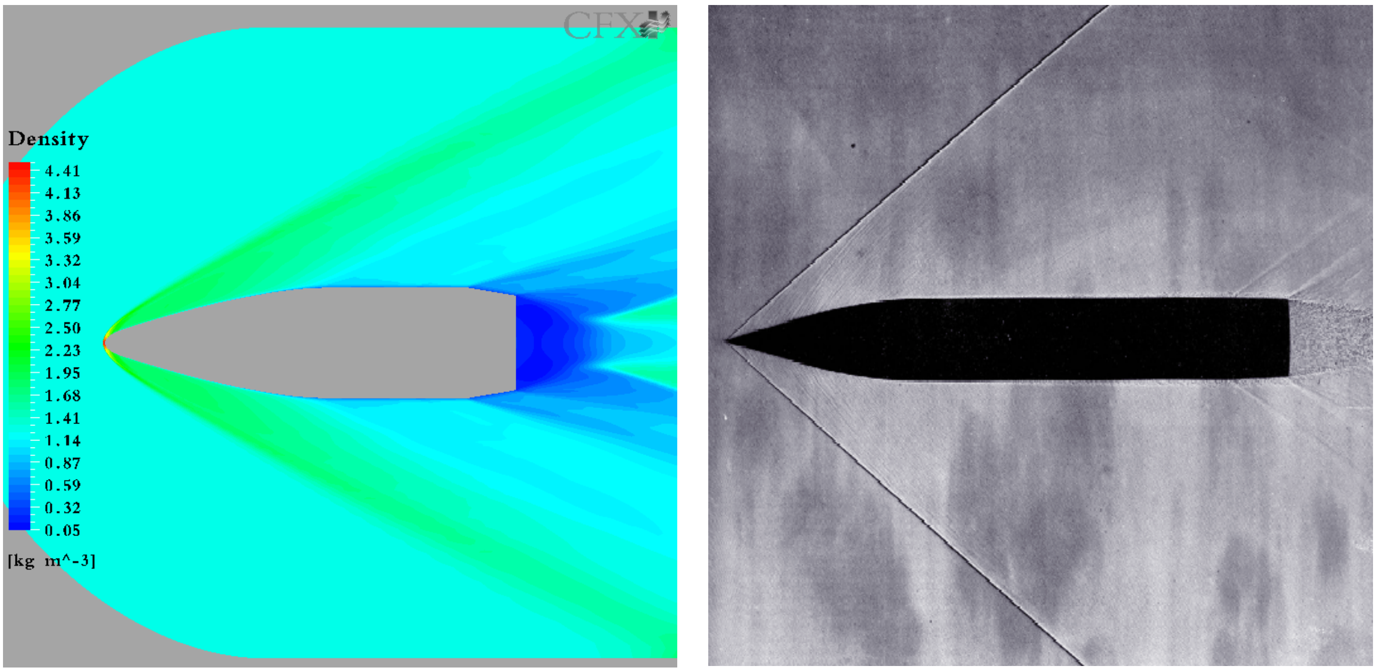

Figure 5. Density distribution in the symmetry plane (left) and shadow photo of supersonic (M=2.58) flow around shell, photo from Transonic Range, U.S. Army Ballistic Research Laboratory (right)

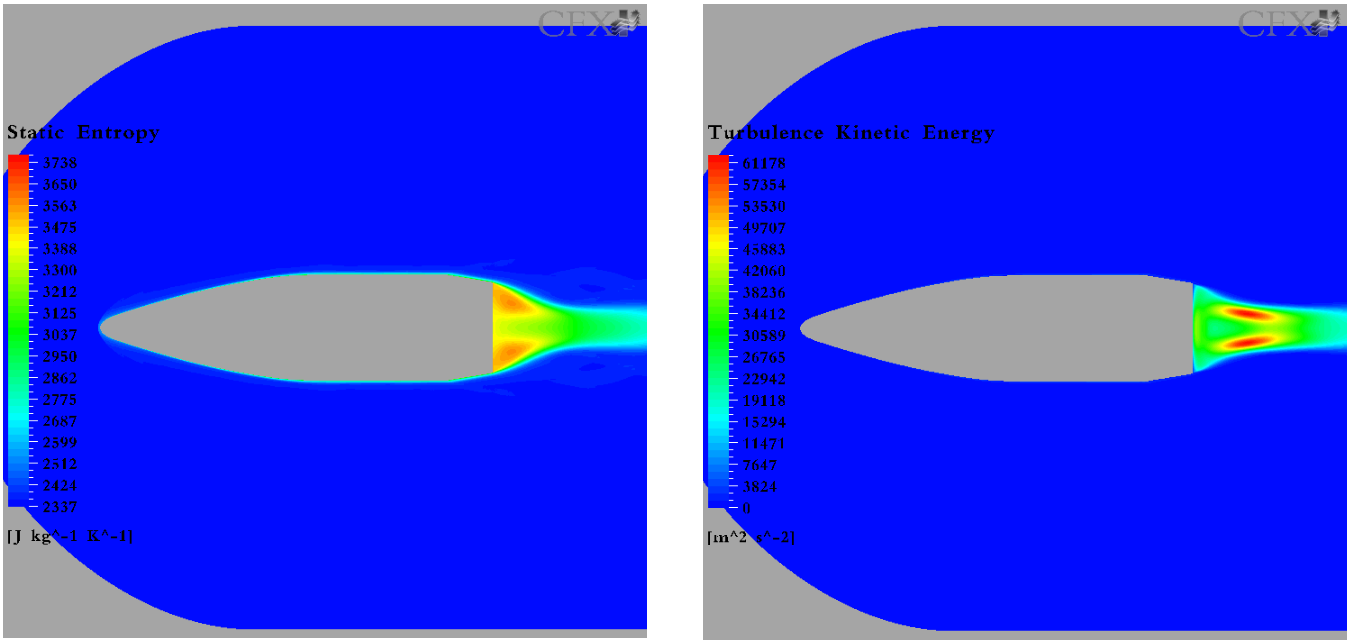

Figure 6. Static entropy distribution (left) and turbulence kinetic energy distribution (right) in the symmetry plane

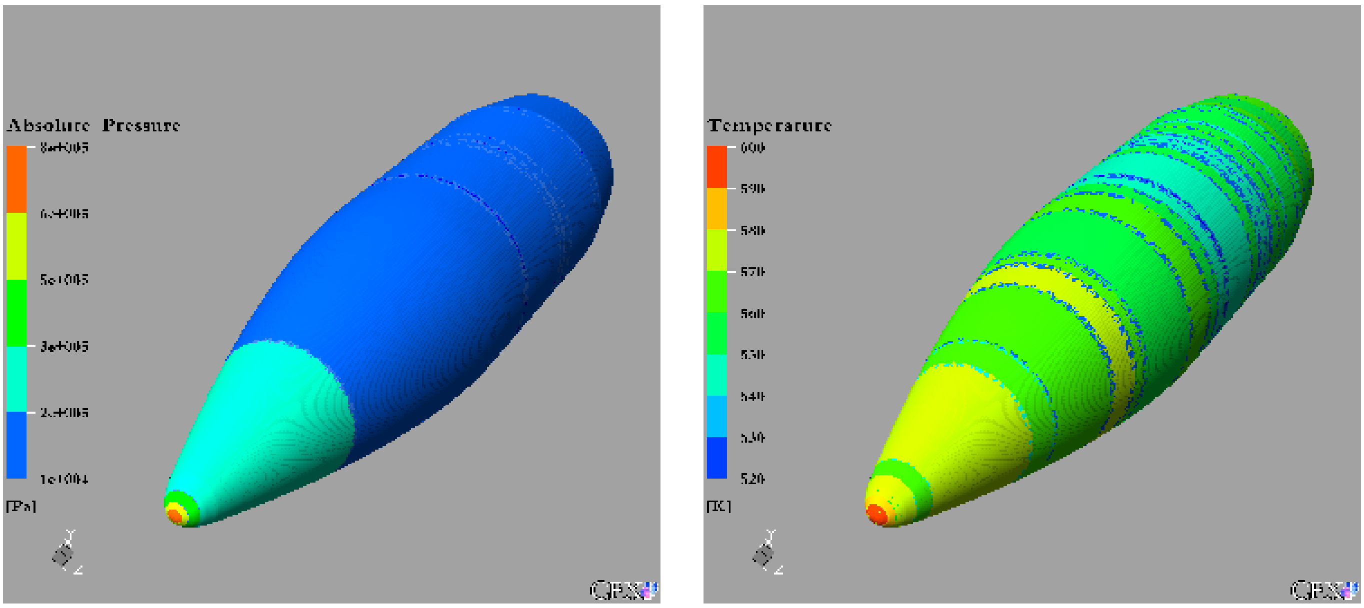

Figure 7. Pressure distribution (left) and temperature distribution (right) at the bullet surface

Numerical calculations of supersonic flow around the bullet in *ANSYS CFX-5.7.1* program carried out on personal computer have shown great possibilities and high accuracy of *CFX* gas dynamic program.

The considered problem of supersonic viscous detached flow about axisymmetric blunt body with flat base, with detached advance shock wave and large separation region behind body base is sufficiently hard task successfully solved by CFD methods.

Numerical solutions allow us reliably calculate qualitative and quantitative characteristics of such flows. For example it will be observed the high temperature level (up to 600°K) at the bullet nose part surface and in the turbulent separation zone at the base region.

As calculation results we have got values of bullet friction drag coefficient $Cd_f=0.0173$ and base drag coefficient $Cd_{base}=0.0112$.