CENTRAL AERОHYDRODYNAMIC INSTITUTE
professor N.E. Zhukovsky

COMPLIANCE OF MODERN METHODS OF NUMERICAL SIMULATION OF HIGH-SPEED FLOWS WITH REAL PHYSICAL PHENOMENA. VALIDATION OF NUMERICAL METHODS

N.V. Voevodenko, A.A. Gubanov, D.S. Ivanyushkin, Yu. G. Shvalev,

TsAGI, Russian Federation

3rd International Symposium on Hypersonic Flight

Air Force Academy (Pozzuoli), Italy, May 30-31, 2019

AAA – Sez. Roma Due “Luigi Broglio”

Main ideas

- The possibilities of modern numerical methods seem almost limitless. Now scientists can numerically model very complex flows for very complex geometries and obtain very beautiful and forceful pictures of flows.
- However, even if we obtained a stable numerical solution that approximates the solution of the original differential equations, it remains an open question whether the resulting solution corresponds to the real physics of the phenomenon under consideration.
- In this regard, there is a need for validation of numerical methods, which is possible only on the basis of a comparison of the obtained numerical results with the data of a physical experiment in wind tunnels or in flight.
- Examples of how stable convergent solutions of the RANS equations do not at all correspond to the physics of the simulated phenomena are given below in this paper.

Main ideas

Central Aerohydrodynamic Institute
n.a. prof. N.E. Zhukovsky

- In aerodynamics, the most frequent inconsistency between numerical solutions and real physics of phenomena is encountered in the numerical simulation of complex separated flows, eddies, and laminar-turbulent transition. Unfortunately, just these phenomena are typical for hypersonic flows.
- Laminar-turbulent transition (LTT), vortex and separation zones are essential features of the flow around the aircraft with hypersonic velocity. Therefore, to the question of the validation of numerical methods must be given special attention.
- Below in two examples, it will be shown how important is the correct simulation the above phenomena when calculating hypersonic flows. It is shown that, using standard approaches, believable flow patterns can be obtained, which, in fact, do not correspond to the physical flow pattern and give fundamentally wrong results. It is possible to detect such a discrepancy only by comparing the results of calculations and experiment.

Examples from HEXAFLY-INT project

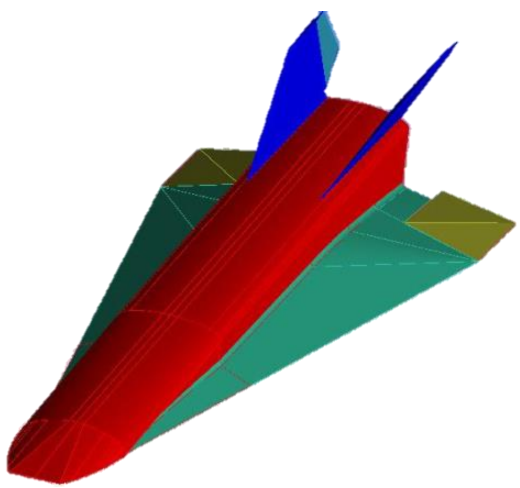
Both examples refer to the research the model of a high-speed passenger aircraft with a cruising Mach number $M_\infty = 7 \div 8$. These studies were conducted in the framework of the international project HEXAFLY-INT with the participation of the EU, Russia and Australia.

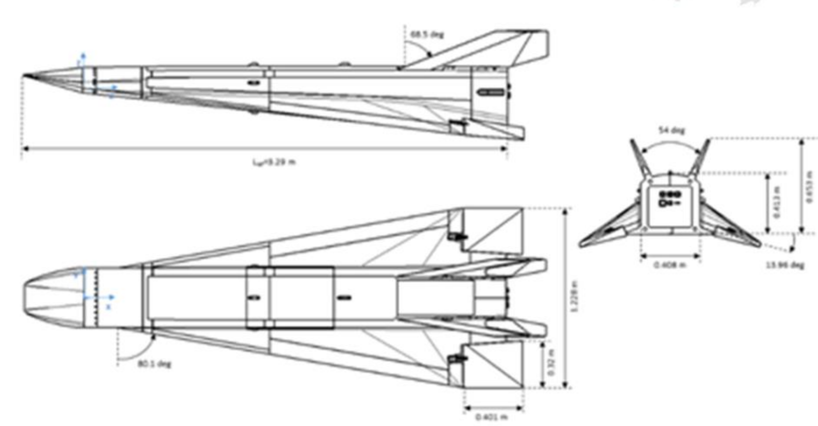
Example 1 – HEXAFLY-INT Glider

Example 1 relates to studies of the external aerodynamics of an experimental aircraft EFTV (Experimental Flight Test Vehicle) without an engine of the so-called “glider” [1]. Usually in numerical simulation based on the solution of the RANS equations, these equations are closed by one of the turbulence models, assuming that the flow occurs with a completely turbulent boundary layer. However, in the case of hypersonic flows, this is not always right.

Powered Consept Model Geometry:

HEXAFLY-INT

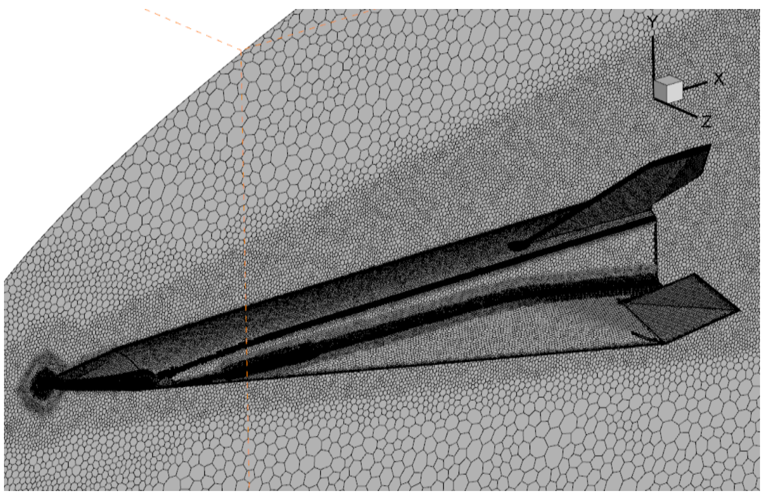


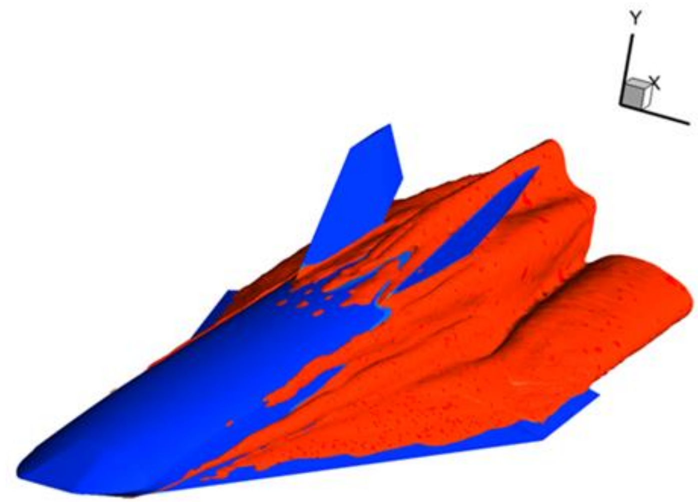


N.V. Voevodenko, A.A. Gubanov, D.S. Ivanyushkin, Y.G. Shvalev, J. Steelant, ‘*CFD and Experimental Simulation of the Laminar-Turbulent Transition on the HEXAFLY-INT Glider Mode’,* 7th European Conference for Aeronautics and Space Sciences (EUCASS), Milan, Italy, 3-6 July 2017: ID 419.

CFD modeling

- The ANSYS FLUENT software package was used for numerical simulation, which was carried out on the basis of the solution of the Reynolds-Averaged Navier – Stokes equations (RANS). To simulate an LTT, we used the SST γ-Re_θ Lentry-Menter turbulence model (Langtry & Menter SST - Transition Model). The calculations used the standard settings of this model, specified in the FLUENT package. A numerical simulation was carried out for the modes corresponding to the points of the flight path of the "glider" (M_∞ = 7., 7.5, α = -5 °, 0, 3.6 °, 15 °, Re≈3. ÷ 10. · 10^6), and the results were analyzed for an LTT on its surface, obtained using the ANSYS FLUENT package.
- The dimension of the computational grid was about 20 000 000 cells for the half model.
- No-slip boundary conditions were met on a solid surface. The condition of radiation from aerodynamic heating of the surface were set on the model surface. On the surface of the elevons, the emissivity corresponded to the value of ε = 0.8, and on the rest of the body to the value of ε = 0.4. To achieve convergence and obtain a steady-state solution, an average of 10,000 to 15,000 iterations was required.

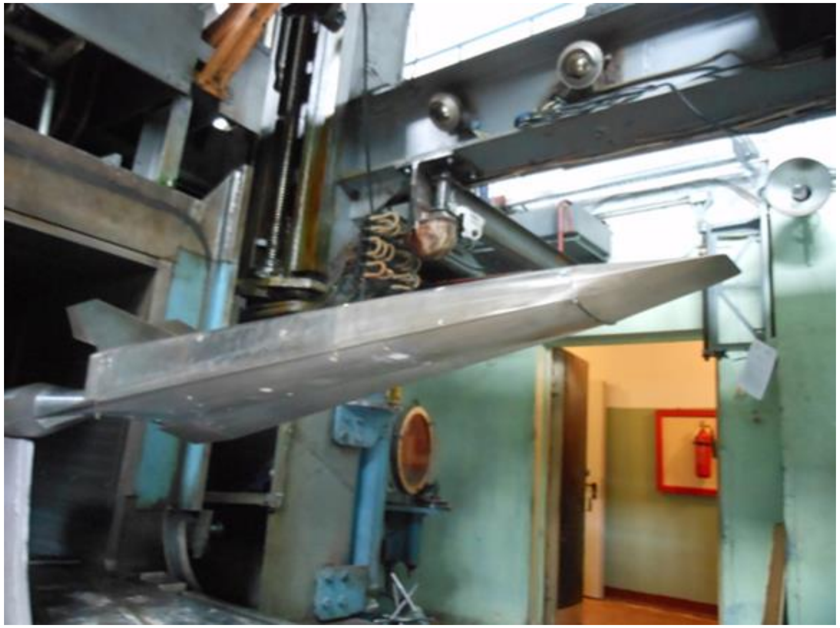




[1] N.V. Voevodenko, A.A. Gubanov, D.S. Ivanyushkin, Y.G. Shvalev, J. Steelant, '*CFD and Experimental Simulation of the Laminar-Turbulent Transition on the HEXAFLY-INT Glider Mode'*, 7th European Conference for Aeronautics and Space Sciences (EUCASS), Milan, Italy, 3-6 July 2017: ID 419.

LTT Experimental studies in TsAGI's wind tunnel T-116

Central Aerohydrodynamic Institute
n.a. prof. N.E. Zhukovsky



The "glider" EFTV HEXAFLY-INT model was manufactured in TsAGI on a scale of 0.35, and tested in a supersonic and hypersonic wind tunnel of TsAGI T-116.

Tests of the EFTV Glider Model at TsAGI T-116 Wind-Tunnel

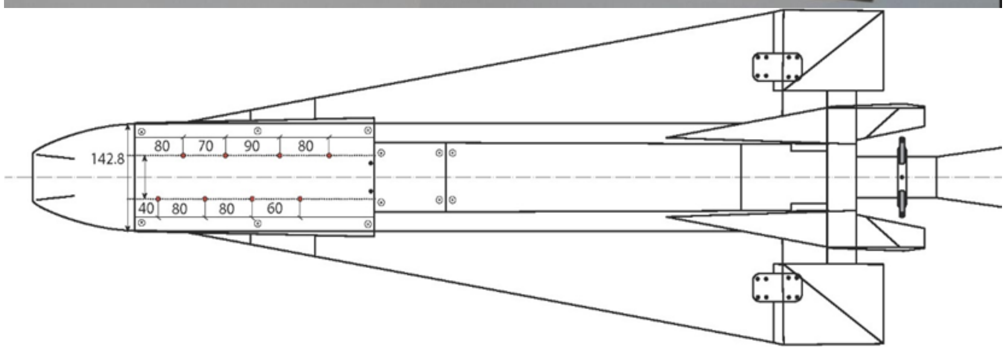
ЦАГИ
Central Aerohydrodynamic Institute
n.a. prof. N.E. Zhukovsky

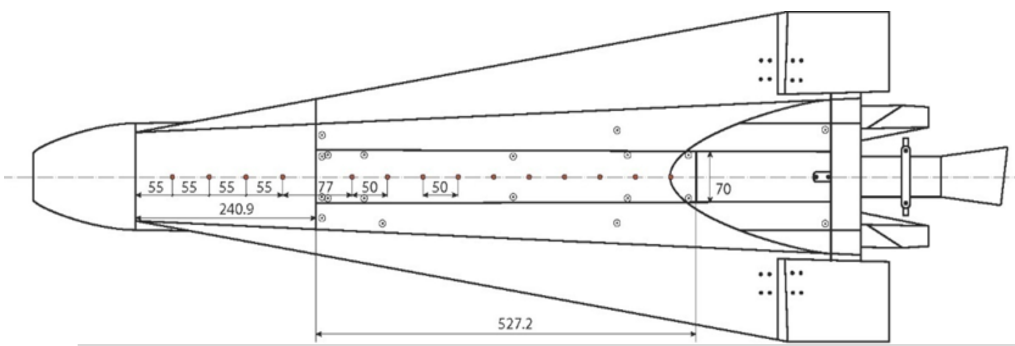
Main feachers T-116

- Mach Numbers: $\mathbf{M_\infty = 1.8 \div 9.8}$
- Reynolds Number range: $\mathbf{Re_{L=1m} = 4.0\times10^6 \div 47\times10^6}$
- Stagnation Pressure range: $\mathbf{P_0 = 1.75\times10^5 \div 80\times10^5}$ **Pa**
- Stagnation Temperature range: $\mathbf{T_0 = 300 \div 1075}$**K**
- Working gas is **air**
- Operating time: **up to 7 minutes**
- Test section dimensions: **2.35 m × 1 m × 1 m**
- Profiled nozzle length : **L = 5.0 m**
- Nozzle exit dimensions: **1m×1m (M≤4.0); D=1.0m (M≥5.0)**
- Mechanism of introducing/removing the model
- Angle of attack range: $\boldsymbol{\alpha = -6° \div 30°}$ **or** $\mathbf{24° \div 60°}$
- Sideslip angle range: $\boldsymbol{\beta = -4° \div 9°}$

Central Aerohydrodynamic Institute
n.a. prof. N.E. Zhukovsky

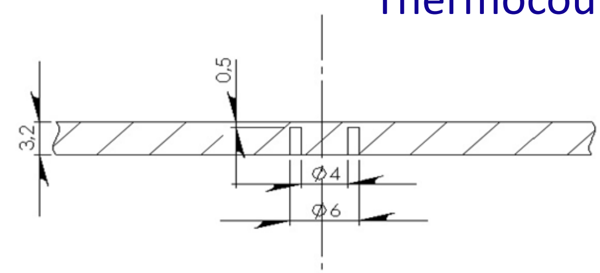
M	P_{tot}, atm	T_{tot}, K	Re_{1M}*10^{-6}	Simulated flight altitude H, km
6.99	22	675	7.66	30





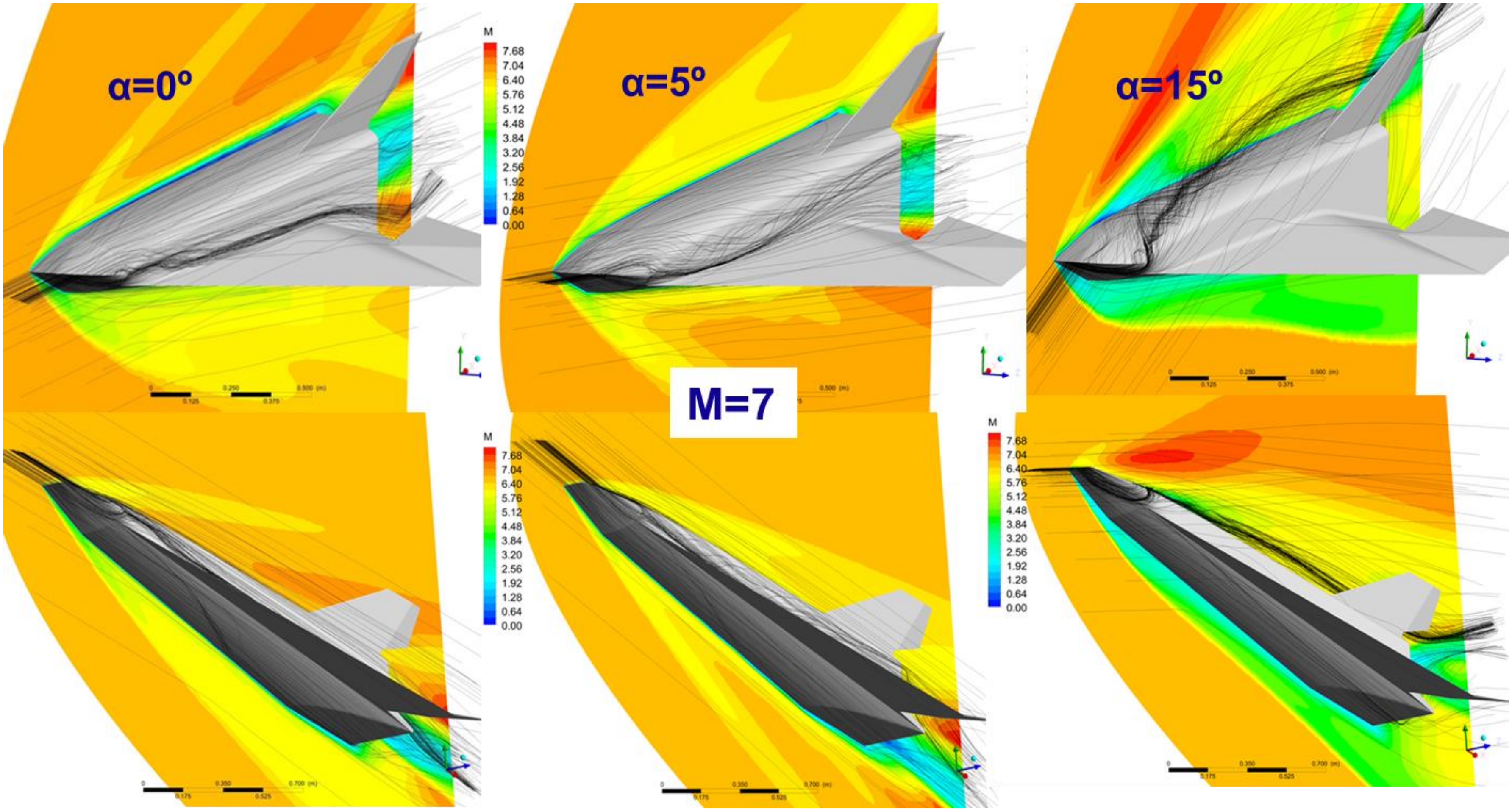
The transition region on the model surface was determined by the thermal method from the values of the heat transfer coefficient (Stanton number), which was determined using special thermal sensors. The determination of the transition region is based on a significant difference in the Stanton numbers in the laminar and turbulent boundary layers.

Thermocouple



LTT CFD studies (Tran1) at M_∞=7 – Mach number field.

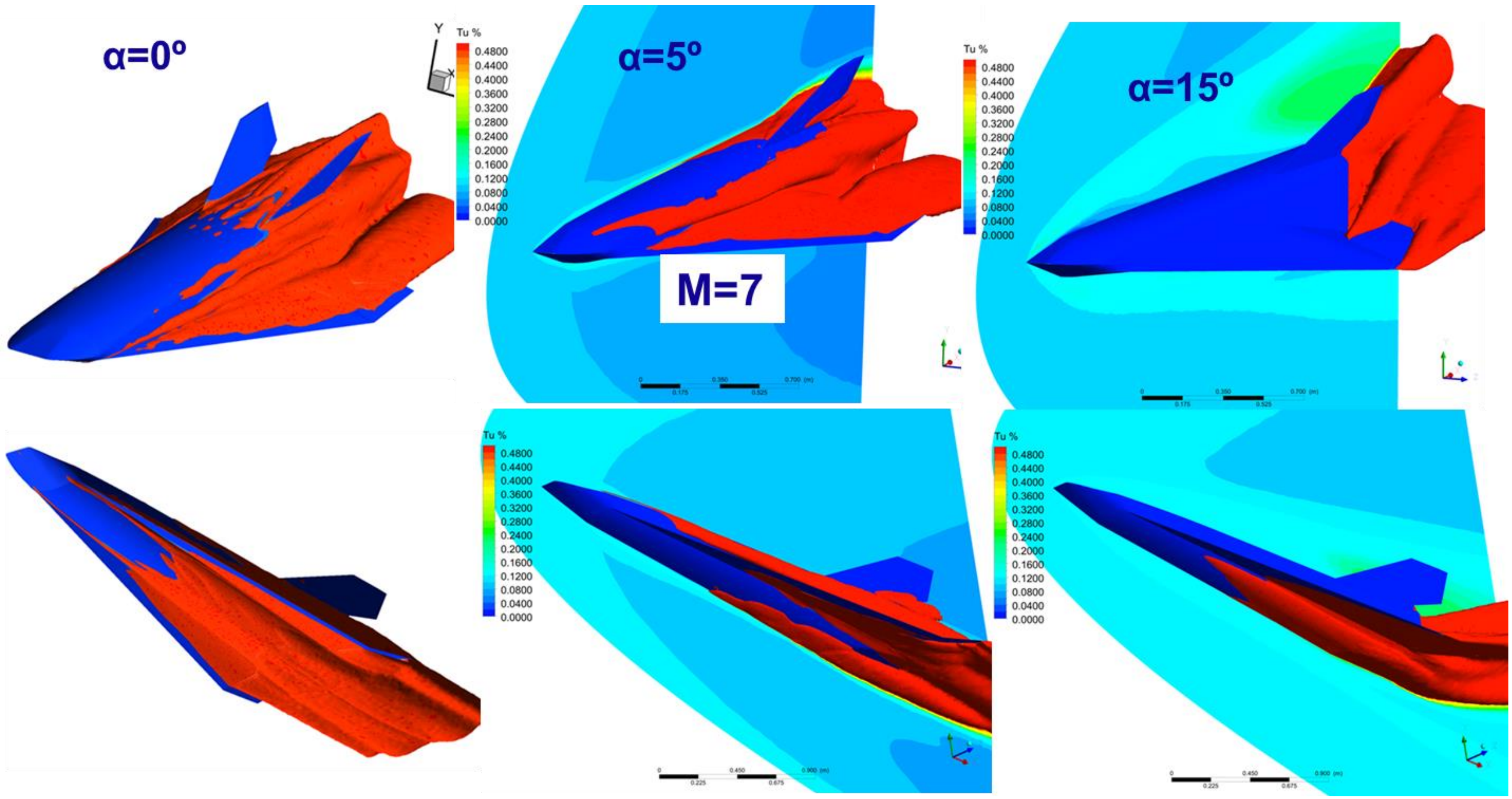
Central Aerohydrodynamic Institute
n.a. prof. N.E. Zhukovsky



CFD simulation for HEXAFLY-INT glider.

Central Aerohydrodynamic Institute
n.a. prof. N.E. Zhukovsky

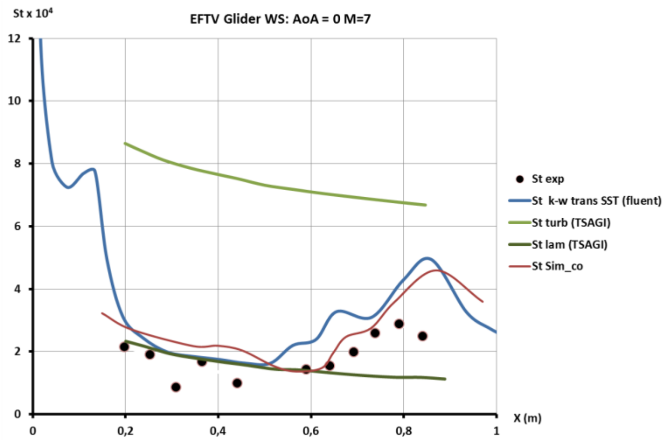
Simulation with LTT(Tran1) at M_∞=7 and α=0° : blue color - Tu < 0.5% (laminar flow), red color - Tu ≥ 0.5% (turbulent flow).

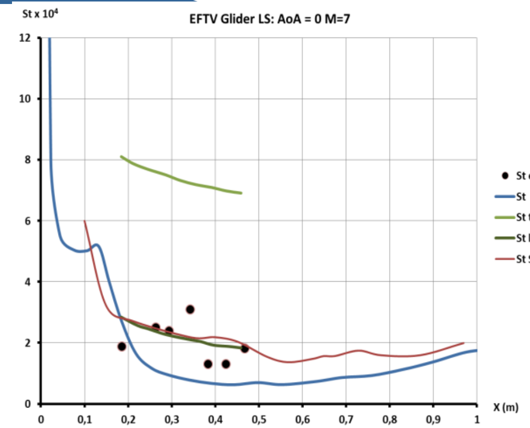


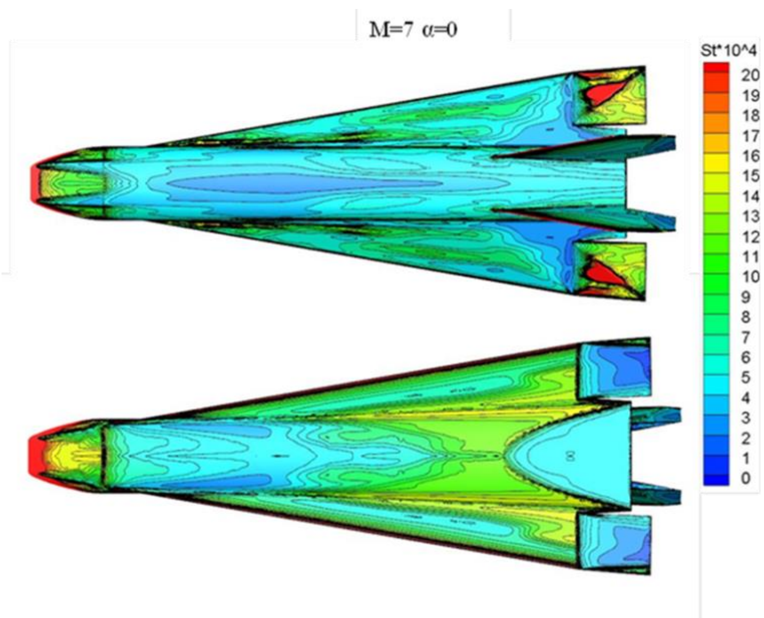
$$Tu = \frac{\sqrt{2/3 k_\infty}}{U_\infty} \cdot 100\%$$ - turbulence intensity

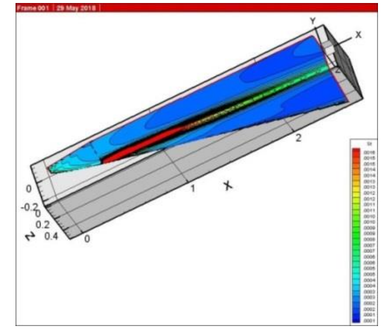
LTT on HEXAFLY-INT glider model M=7 AoA=0

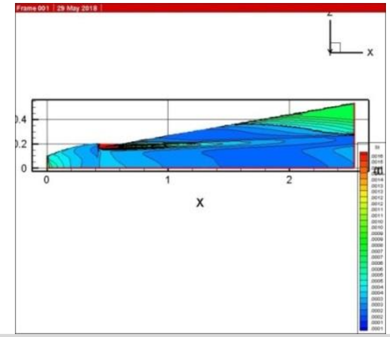
Central Aerohydrodynamic Institute
n.a. prof. N.E. Zhukovsky









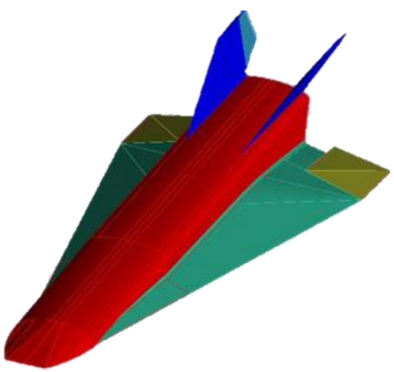


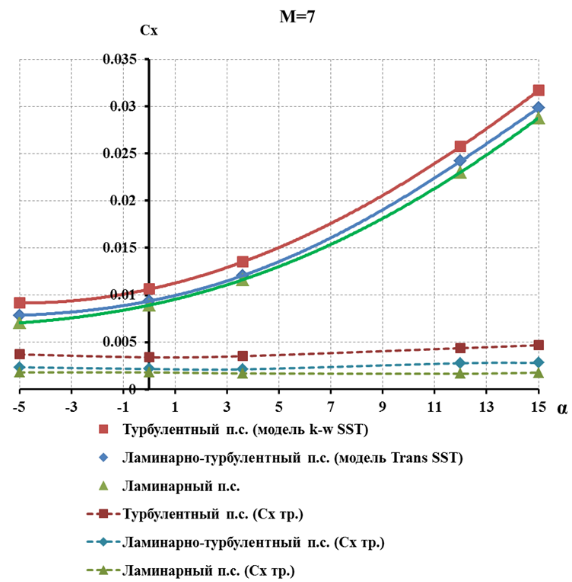
LTT influence on the HEXAFLY-INT glider aerodynamic coefficients.

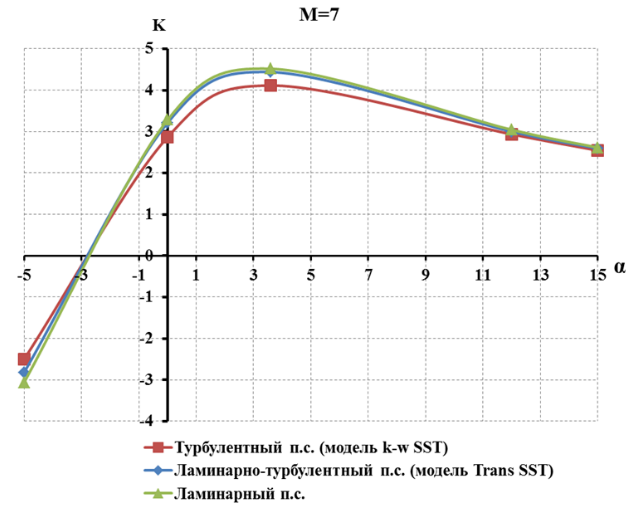
Central Aerohydrodynamic Institute
n.a. prof. N.E. Zhukovsky

Conducted computational and experimental studies of HEXAFLY-INT glider model have shown that:

- Significant part of the model surface is flown around with laminar BL (from 30 to 70% and high with Mach growing up) or transitional BL;
- LTT has a significant impact on the total aerodynamic characteristics of the aircraft. The contribution of the friction drag to the total vehicle drag at M = 7 and zero angle of attack with a fully turbulent BL is 30%, with a fully laminar BL - 17%, and with flow with LTT it does not exceed 19% of total model drag;
- The loss of the lift-to-drag racio of the vehicle as a result of the viscosity effect with turbulent BL is approximately 0.5 of the units ($L/D_{max} \approx 4$ is reached in the range of angles of attack α = 2 ° - 2.5 °), which corresponds to 12% L/D_{max}.







Example 1 – Conclusions:

1. In the majority of considered modes on the windward side of the model, LTT begins approximately in the middle of the model length and ends after its end. On the leeward side of the model, in most cases, the flow is laminar.

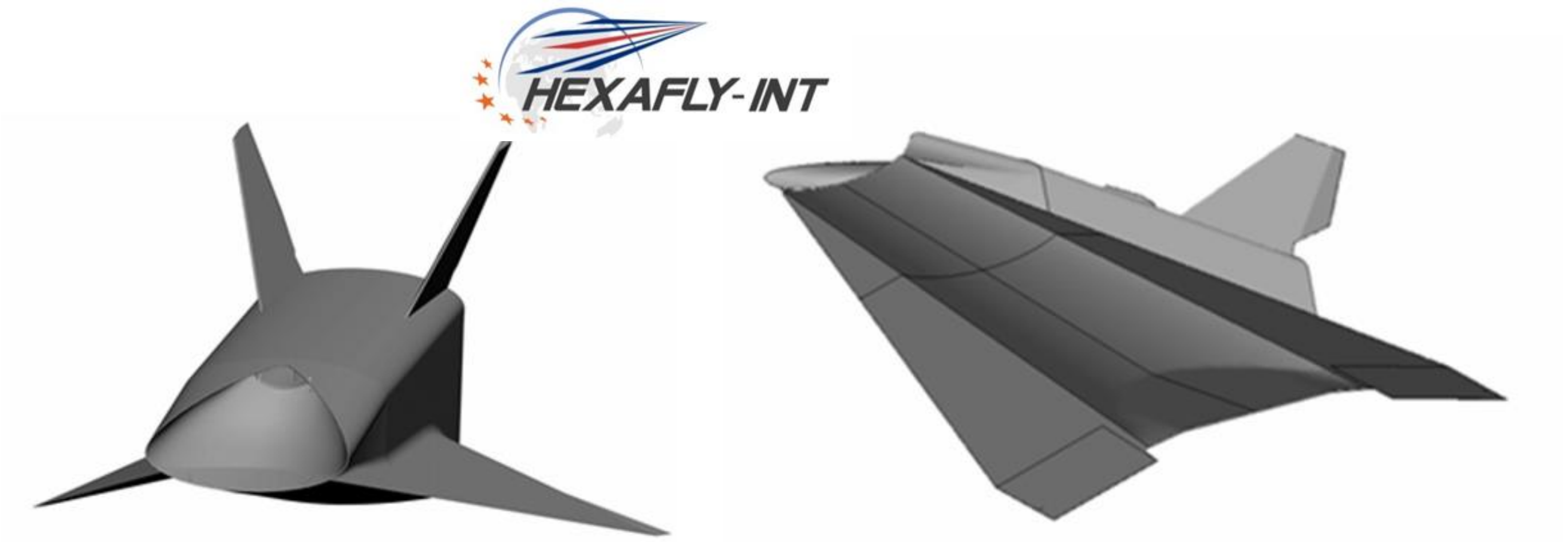
2. A significant influence on the flow pattern on the model leeward side is done by complex vortex structures descending from the nose and wing leading edges. They initiate transitional phenomena downstream.

3. **In general, the flow around the glider model HEXAFLY-INT is a laminar-turbulent mixed, therefore it is necessary to use computational methods with LTT modeling on the considered flow regimes of such models.**

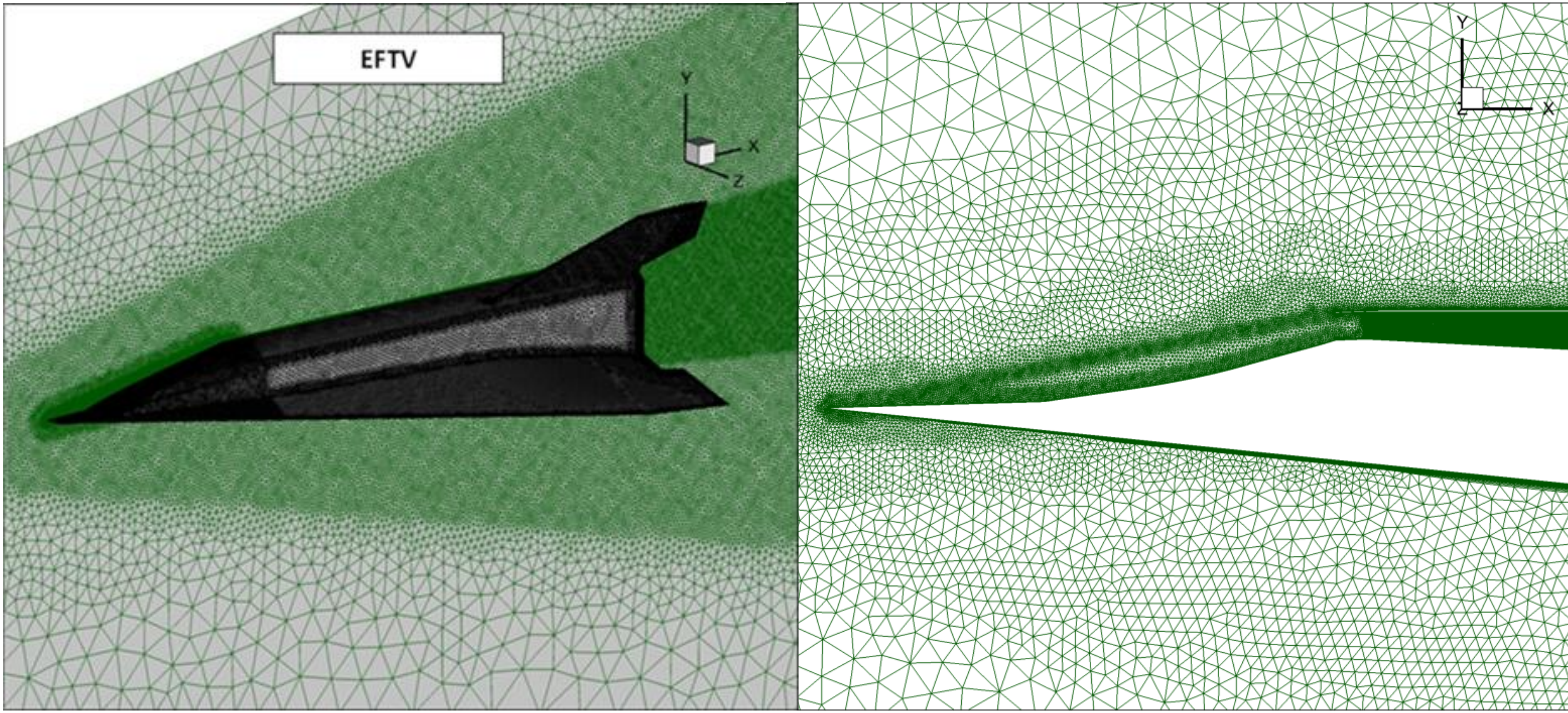
4. **Numerical simulation based on the RANS equations solution with a completely turbulent boundary layer is not correct in this case.**

Example 2 –HEXAFLY-INT propelled model

- Example 2 relates to the studies of HEXAFLY-INT model with engine. A characteristic feature of this model is a convergent air intake, turned upwards and located on the upper side of the model.
- As shown by the numerical and experimental studies of this model, the BL state on the braking surface of the air intake device (AID) of this model has a critical effect on the start of this air intake.



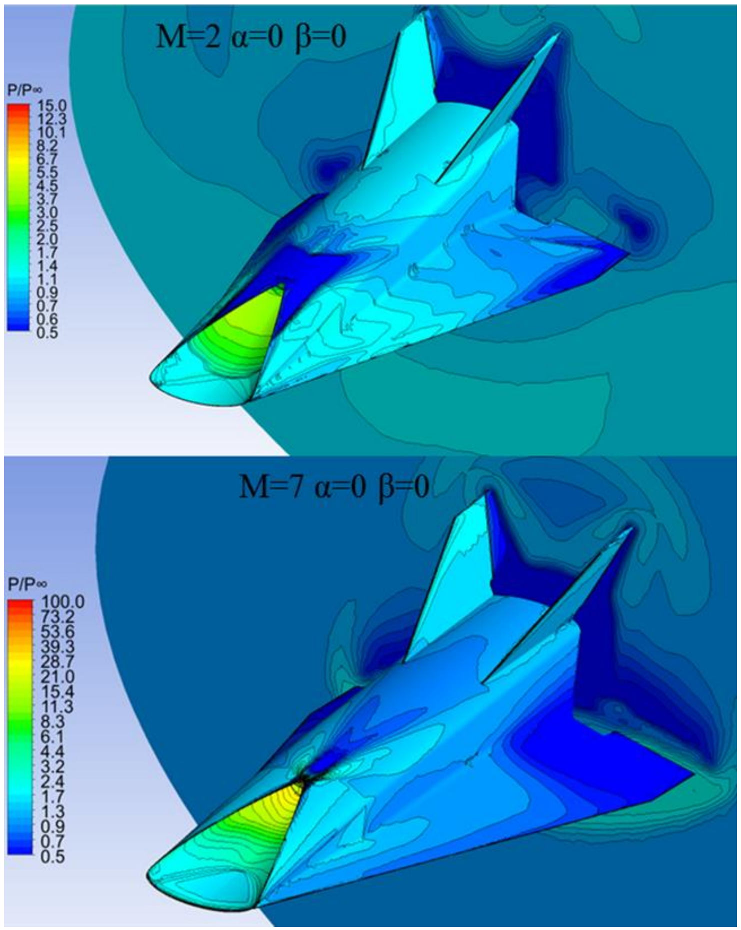
[4] N.V. Voevodenko, A.A. Gubanov, D.Yu. Gusev, M.A. Ivankin, D.S. Ivanyushkin, V.Yu. Lunin, P.A. Meshennikov, Yu.G. Shvalev, V.A. Talyzin and V.A. Yakovleva Boundary layer state influence on start of the inward-turning intake. ICAS-2016-4.10.2 [2016-0383] (30th International Congress of the Aeronautical Sciences, 25-30 September 2016, Daejeon, Korea).



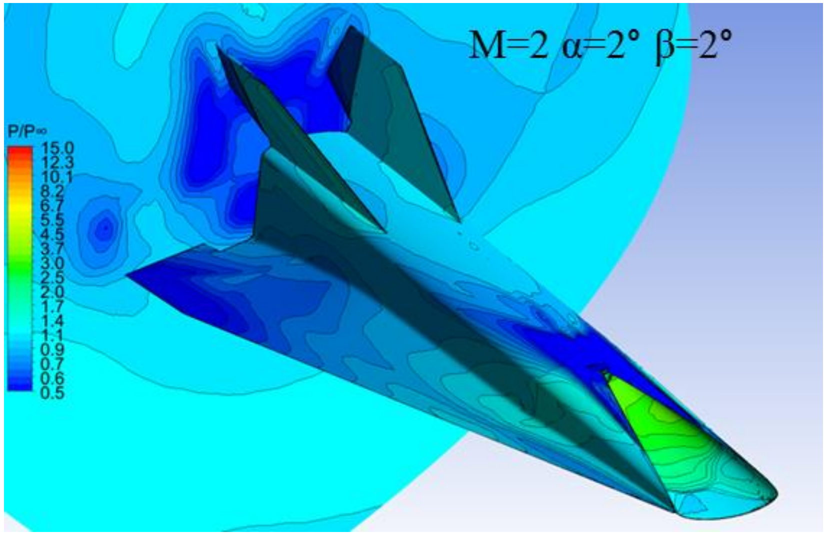
M=7.4 α=0 H=31km
Mesh 40 000 000, Turbulence model - SA

Numerical simulation by FLUENT

Numerical simulation was carried out in the range of Mach numbers M = 2 ÷ 7.5 and angles of attack α = -4 ° ÷ 12 ° at slip angles β = 0 and 2 °. The Reynolds number was calculated on the model length and corresponded to the values:

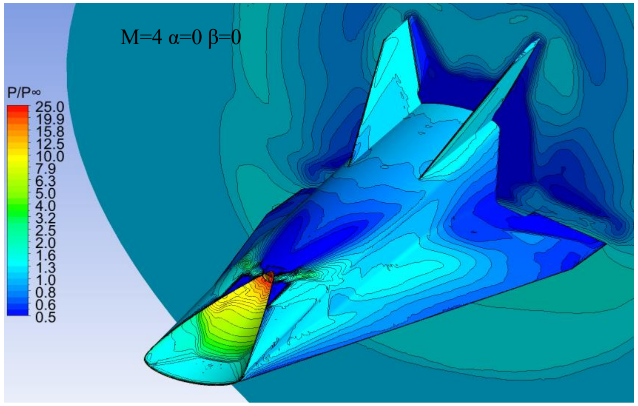


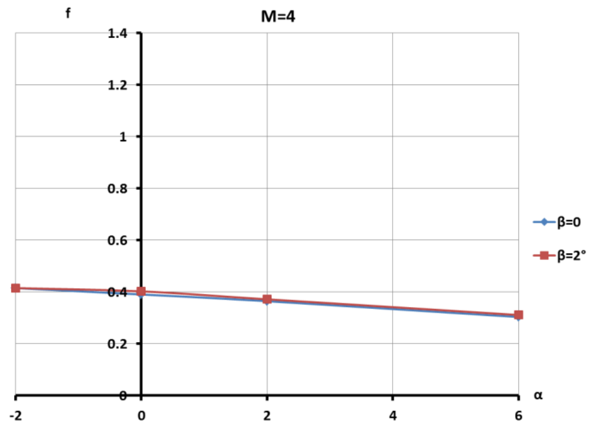
- Re_L=9.97*10^6 for M=2
- Re_L=6.69*10^6 for M=4
- Re_L=9.38*10^6 for M=6
- Re_L=1.06*10^7 for M=7
- Re_L=3.17*10^6 for M=7.5

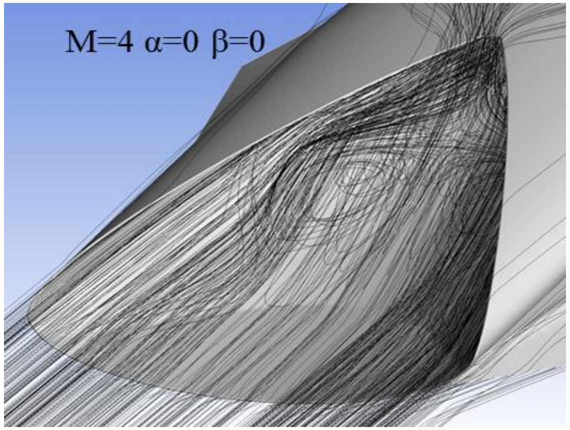


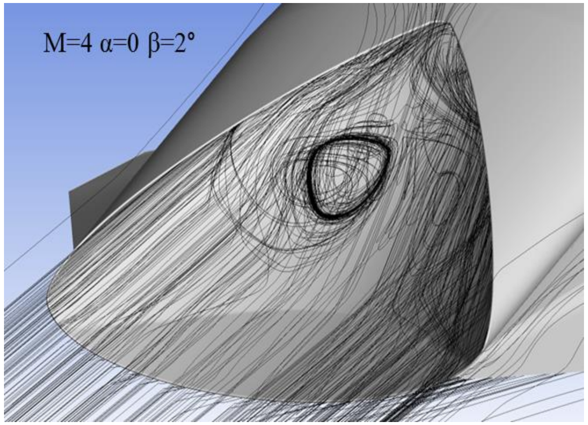
Numerical simulation, M=4

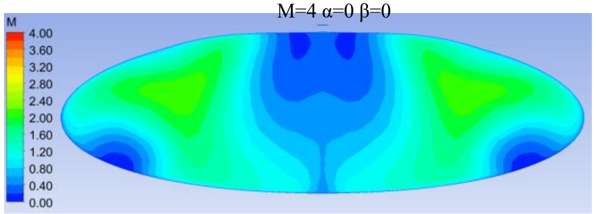
Central Aerohydrodynamic Institute
n.a. prof. N.E. Zhukovsky

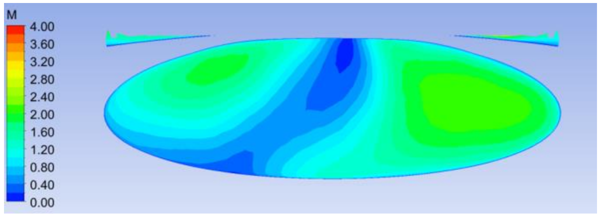


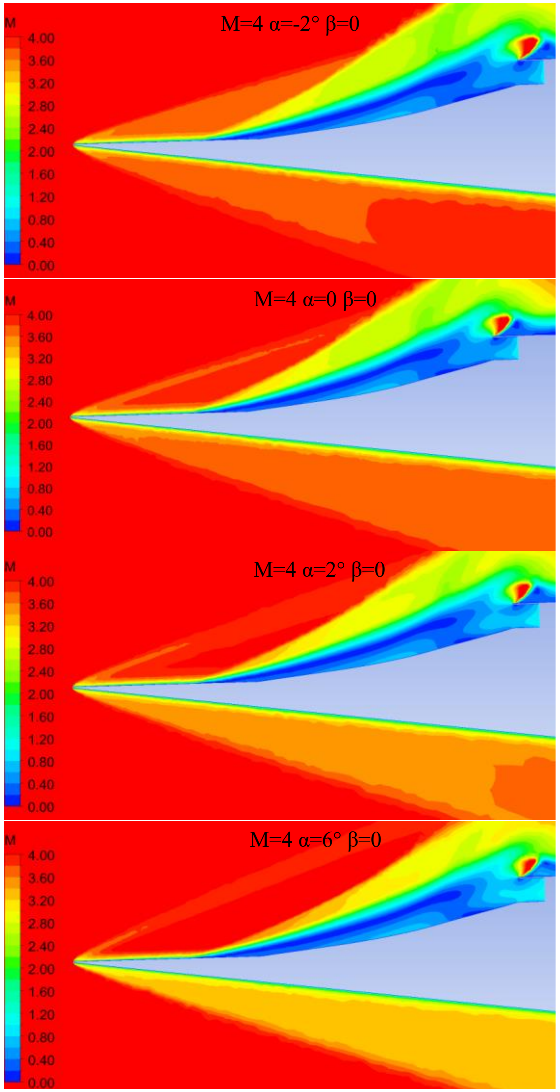






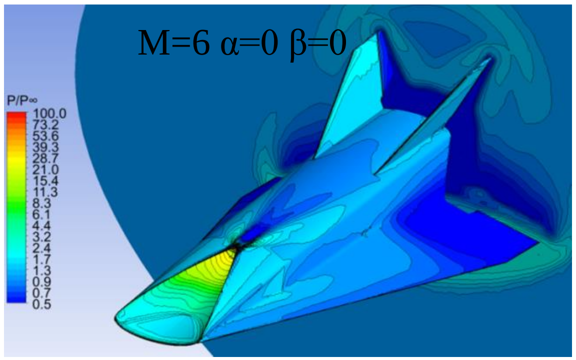


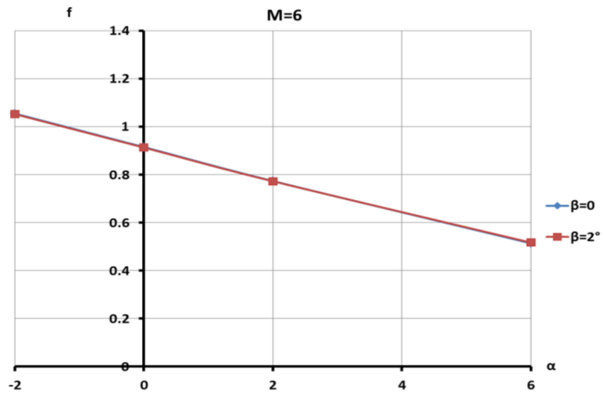


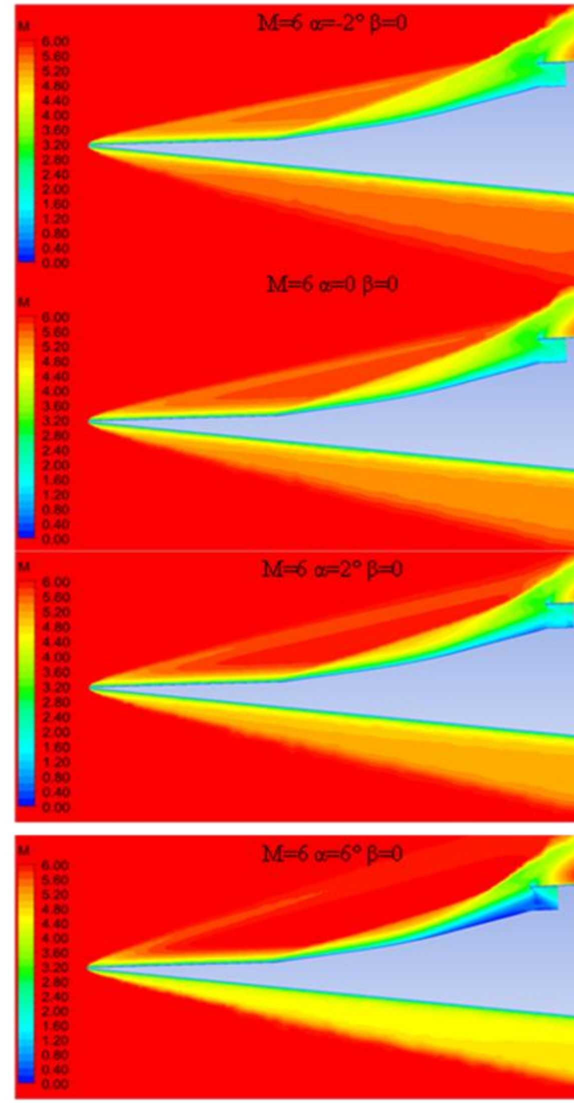


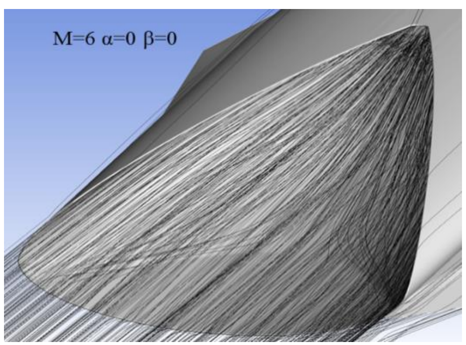
Numerical simulation, M=6

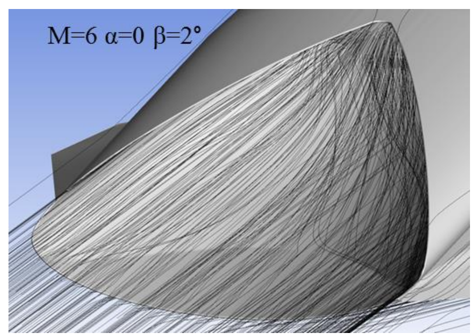
- An increase in the Mach number to M = 6 substantially changes the flow pattern. At angles of attack α = -2 °, 0, the region of subsonic flow is practically absent, and begins to form at positive angles of attack.

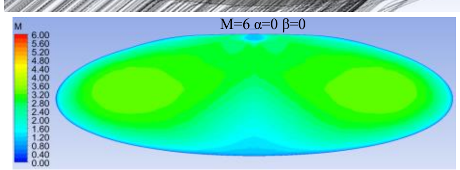


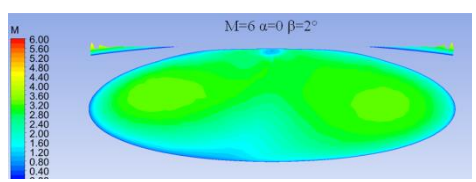








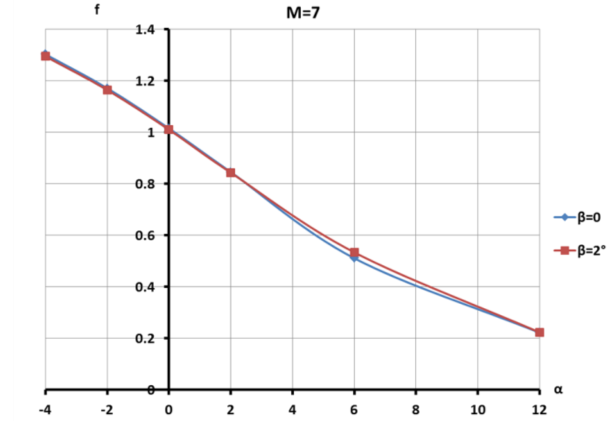


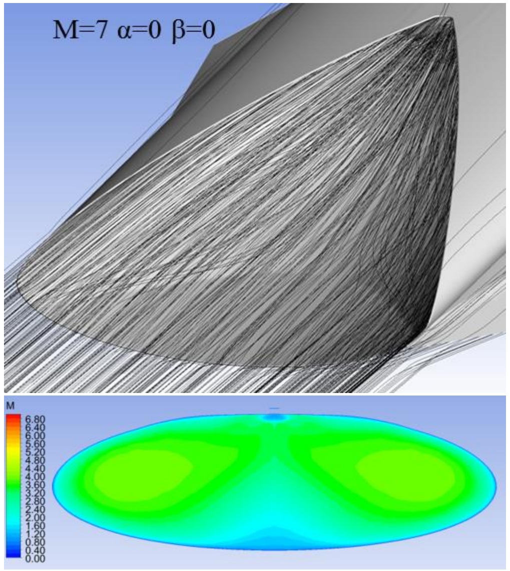


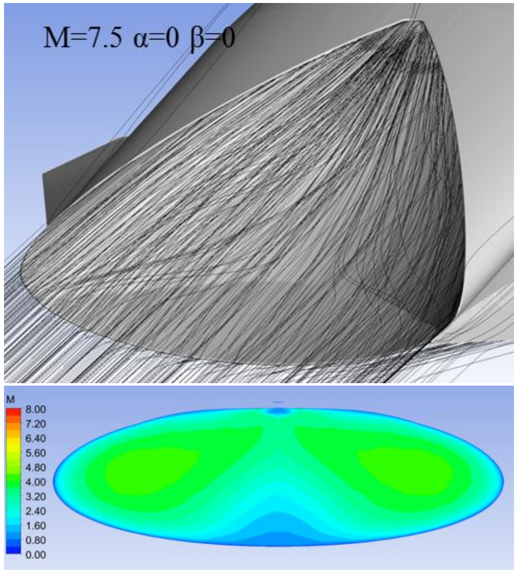
Numerical simulation, M=7 and 7.5

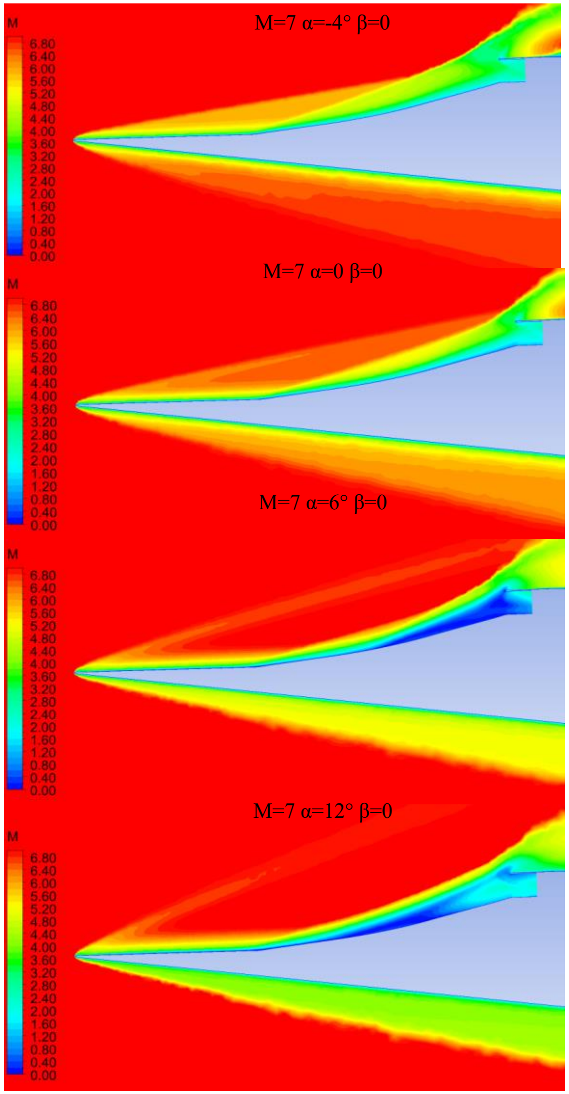
Central Aerohydrodynamic Institute
n.a. prof. N.E. Zhukovsky

- A similar picture of the flow is realized at larger Mach numbers M=7 and 7.5.









Results of numerical simulations with turbulent BL.

- The intake mass flow rate coefficient for Mach numbers M = 2, 4 takes values f = 0.1 ÷ 0.4, which indicates that the air intake device has not been started.

- Analysis of the flow fields showed that the main feature of this intake is the presence of subsonic zone which is formed on the center of the air intake domain practically at all flow regimes. When the Mach number M = 2 ÷ 4 subsonic flow zone is most pronounced and it and extends downstream until the intake entrance.

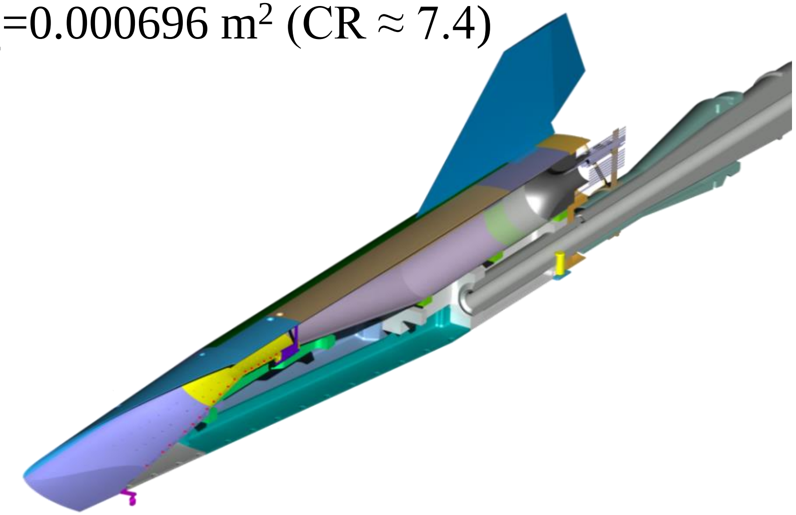
- An increase in the Mach number to M = 6 substantially changes the flow pattern. At angles of attack α = -2 °, 0, the region of subsonic flow is practically absent, and begins to form at positive angles of attack. A similar picture of the flow is realized at larger Mach numbers M=7 and 7.5.

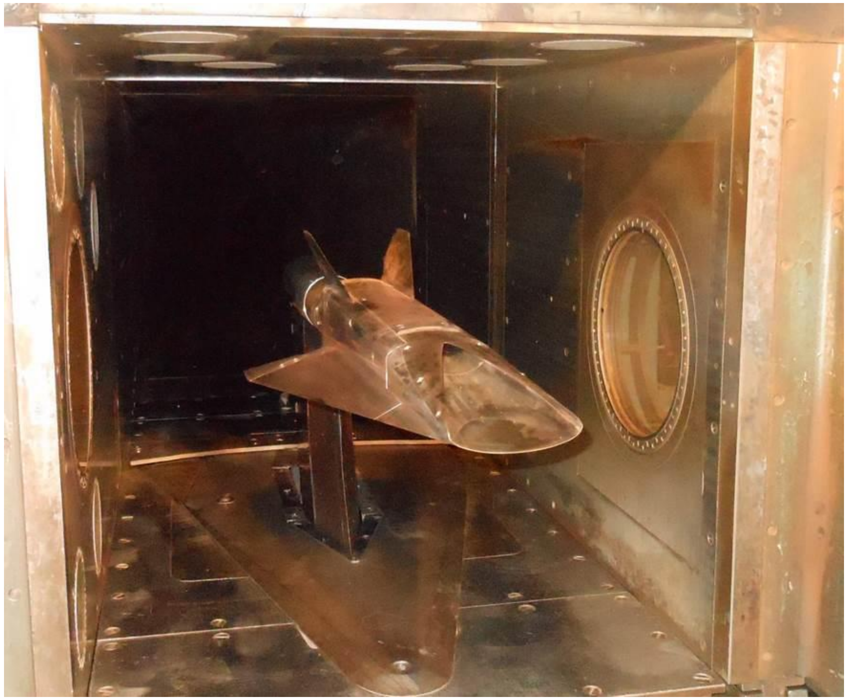
Tests of the EFTV Glider Model at TsAGI T-116 Wind-Tunnel

ЦАГИ
Central Aerohydrodynamic Institute
n.a. prof. N.E. Zhukovsky

The intake was made with 2 throats (yellow inserts), CR - contracting ratio :

$\mathbf{F}_{\mathbf{thr\ 1}}$=0.000595 m^2 (CR ≈ 8.6)
$\mathbf{F}_{\mathbf{thr\ 2}}$=0.000696 m^2 (CR ≈ 7.4)

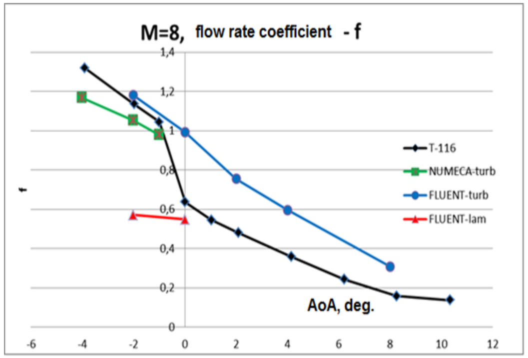


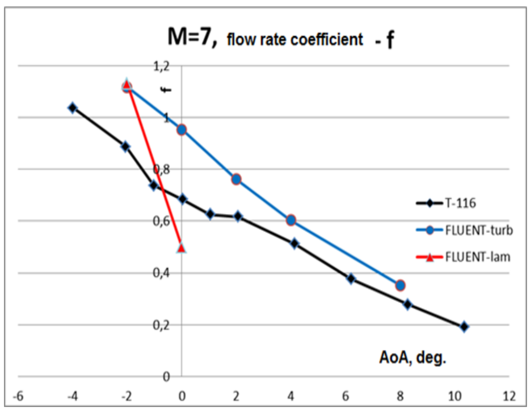


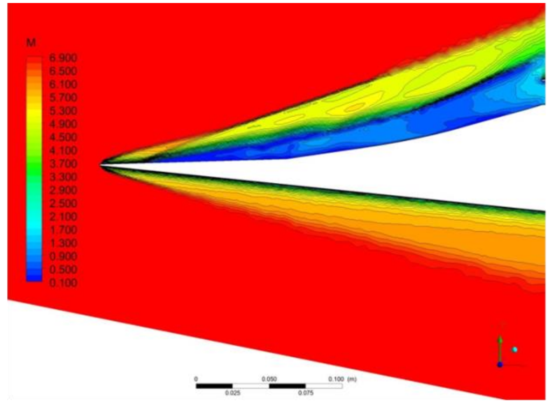
In the wind tunnel T-116, the intake started just with the expanded throat area - CR = 7.4.

Simulation by FLUENT, M_{∞}= 7 AoA=0°, with laminar and turbulent boundary layer.

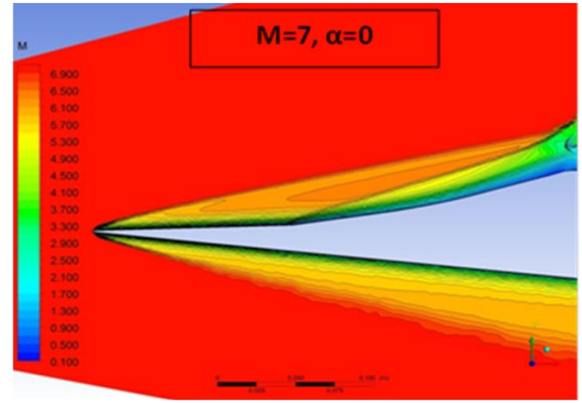
Central Aerohydrodynamic Institute
n.a. prof. N.E. Zhukovsky





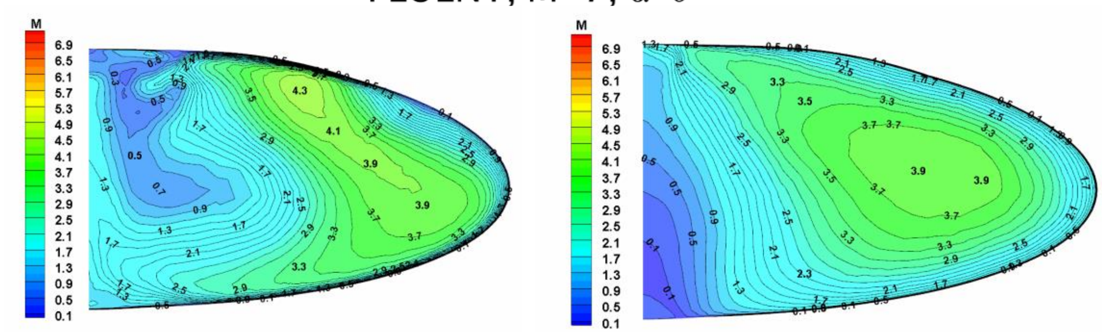


Laminar boundary layer, f≈0.5



Turbulent boundary layer, f=0.95

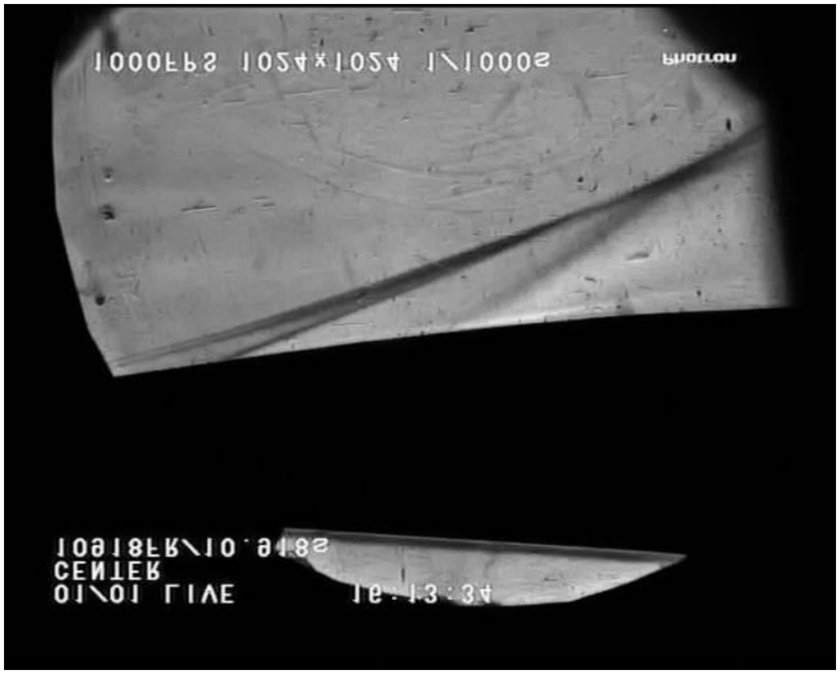
FLUENT, M=7, α=0

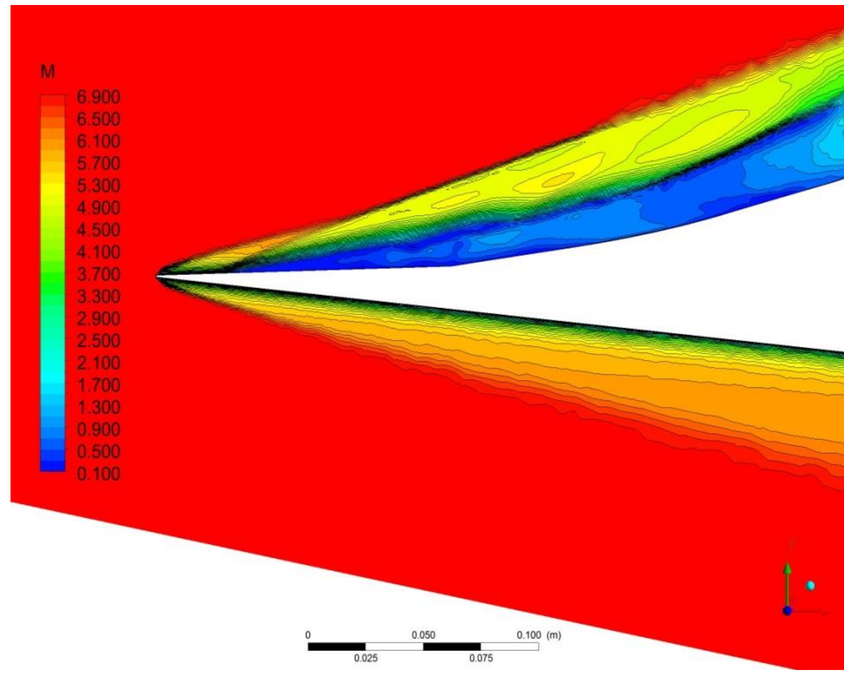


- The results of tests showed that the intake without transition grit started just in very limited range of test flow condition: the intake didn't start at Mach number 7 at all, and starting observed just at Mach number 8 and negative angles-of-attack $\alpha \leq -1°$.

Comparison between experimental and numerical data

M=7, AoA=0



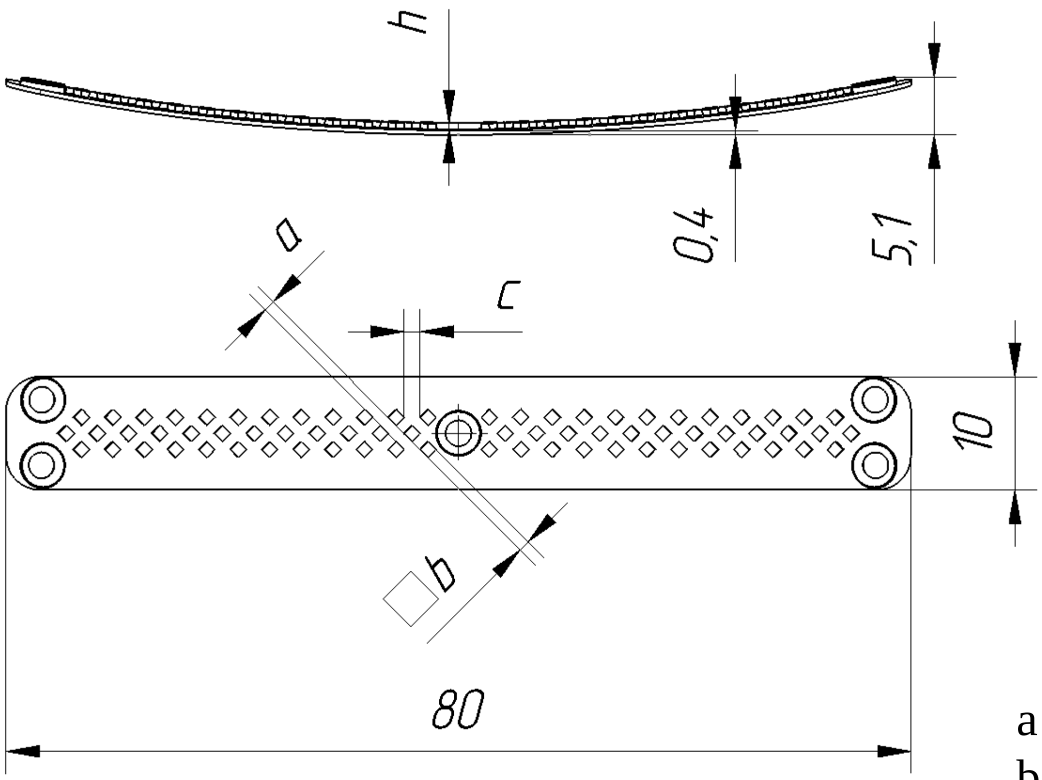


Experimental data
f=0.48-0.5

Numerical data (FLUENT)
Laminar boundary layer,
$f \approx 0.5$

Configuration of BL Tripping Strips, Var. 1

Central Aerohydrodynamic Institute
n.a. prof. N.E. Zhukovsky



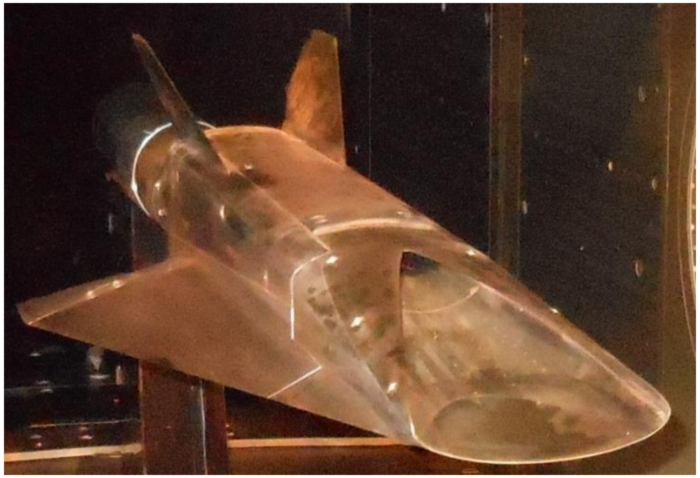
<u>Variant 1 of the BL Transition Grit:</u>
2 metallic strips with 3 rows of diamond-shaped roughness elements each; the heights of the roughness elements were 0.75 mm and 1 mm

a = 1 mm
b = 1 mm
c = 1.4 mm
h = 0.75 mm and
1 mm

Distance from the LE:
10 mm and 30 mm

Aerodynamic tests with BL tripping devices

Central Aerohydrodynamic Institute
n.a. prof. N.E. Zhukovsky



Variant 1

Variant 2

Variant 3

Variant 4

Variant 2: 10 screw heads of a 'dovetail' shape having the height k=1.2 mm and the top diameter D = 3.8 mm installed at distances of approximately 15 mm and 35 mm from the leading edge of the intake at three positions dispersed by the lateral co-ordinate.

Variant 3: the same screw heads with wires of the diameter d = 0.5 mm attached to the model surface by the screws in 'cross' position.

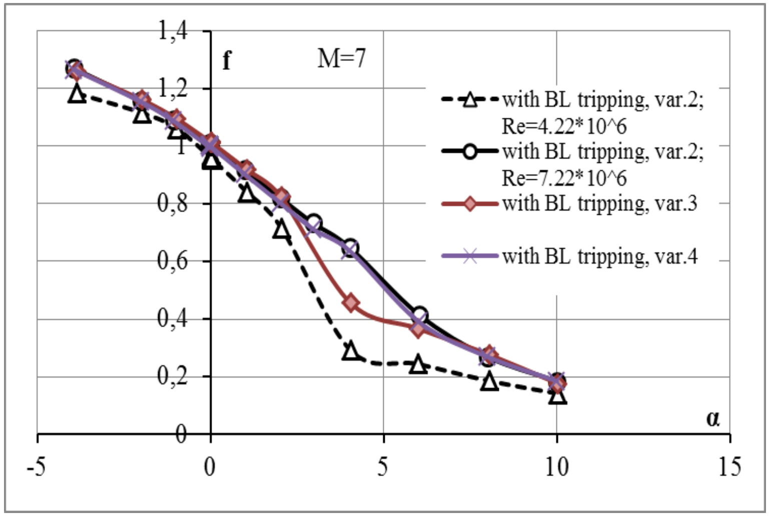
Variant 4: the same screw in 'lines' parallel to the intake leading edge.

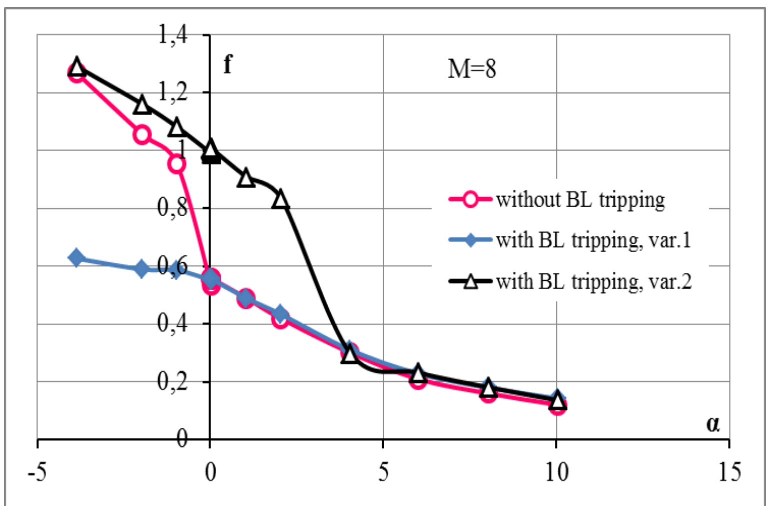
The tests were provided at Mach numbers 7 and 8.

Aerodynamic tests with BL tripping devices

Central Aerohydrodynamic Institute
n.a. prof. N.E. Zhukovsky

Air mass flow coefficient





- Transition grit variant 1 didn’t improve the situation, and the grit variant 2 consisting of 10 screw heads widened the range of the intake starting up to positive angles-of-attack $\alpha \leq 2°$.
- Additional wires in compositions of the transition grits, variants 3 and 4, did not show positive effect on the intake starting.

Variant 1

Variant 2

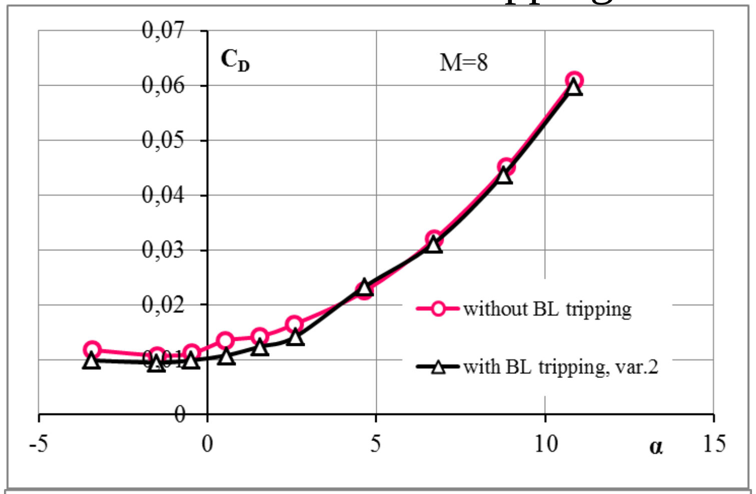
Variant 3

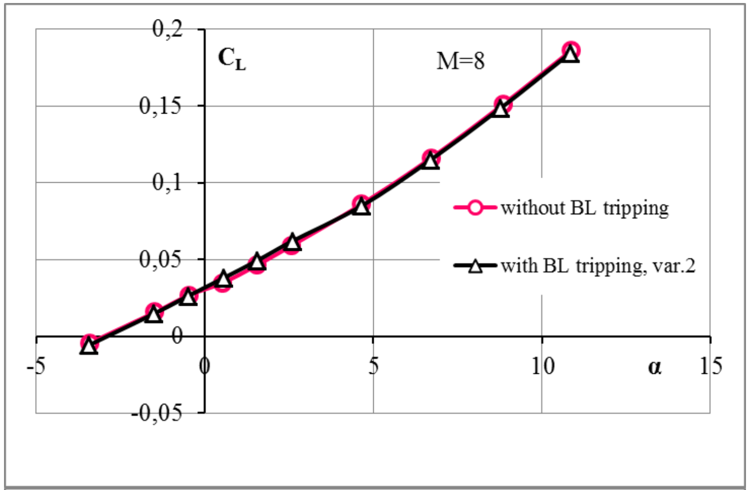
Variant 4

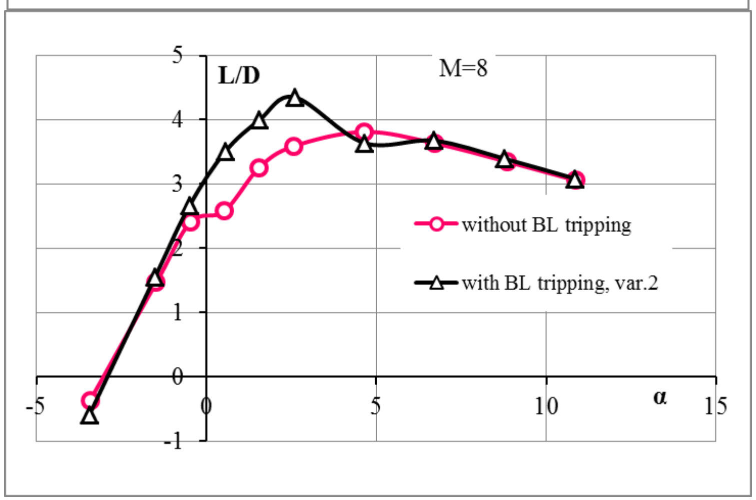
EFTV Powered Aerodynamics: M=8, δ_{flaps}=0

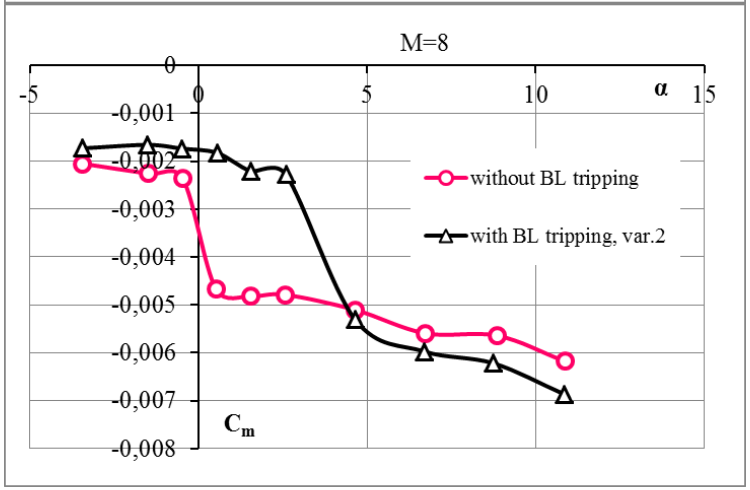
Central Aerohydrodynamic Institute
n.a. prof. N.E. Zhukovsky

The experimental results demonstrate that the maximum value of aerodynamic efficiency L/D of the model with BL tripping is about 4.5.









Example 2 - Conclusions

1. Conducted CFD and EFD studies of the flow in the area of the air intake of HEXAFLY-INT model have shown that one of main feature of this configuration is the occurrence of subsonic separation zone in the intake central part of flow. This feature influences both on the intake performances and on external aerodynamics.
2. The results of the wind tunnel tests showed that start of the intake depends on both the intake contracting ratio CR and the boundary layer (BL) state on the intake surface. In the wind tunnel T-116, the intake started just with the expanded throat area - CR = 7.4.
3. Installation of the transition grit generating a number of vortices near the intake compression surface promoted early BL transition and significantly improved the intake starting performance.
4. Maximum value of aerodynamic efficiency L/D of the model is about 4.5. Unstart of the intake will lead to significant change in L/D (less than 4.) and in the pitching moment coefficient especially at low angles-of-attack.
5. **Numerical simulation based on the RANS equations solution with a completely turbulent boundary layer is not correct in this case and leads to the fundamentally wrong results.**

Main conclusion

The above examples show how important is the experimental validation of numerical results!

Thank You!