11th. World Congress on Computational Mechanics (WCCM XI)
5th. European Conference on Computational Mechanics (ECCM V)
6th. European Conference on Computational Fluid Dynamics (ECFD VI)
July 20 - 25, 2014, Barcelona, Spain

SIMULATION OF TURBULENT FLOW AROUND WEDGE-SHAPED BODY WITH BACKWARD STEP USING IDDES APPROACH ON UNSTRUCTURED MESH

Boris N. Dankov[1], Alexey P. Duben[2]*, and Tatiana K. Kozubskaya[2]

[1] TSNIIMASH, Pionerskaya str., 4, Korolyov, Moscow region, Russian Federation, 141070,
dankovboris@gmail.com, http:// tsniimash.ru
[2] M.V. Keldysh Institute of Applied Mathematics of RAS, Miusskaya Sq., 4, Moscow, Russian
Federation, 125047, dualks@gmail.com, http://caa.imamod.ru/

Key Words: *Hybrid RANS/LES, IDDES, Grey Area, Unstructured Meshes.*

The problem of flow around the wedge-shaped body with a three-dimensional backward-facing step at transonic flow regime is considered (the model is presented at Fig. 1 (left), height of the step is 0.0838L). The characteristic Reynolds number of the problem based on the length of the wall upstream the backward step $\mathrm{Re} = 7.239 \cdot 10^6$, the Mach number $M = 0.913$. The unstructured mesh used in the simulation contains 16M nodes and 92M tetrahedrons. The computation was performed using the IDDES [1] approach and in-house code NOISEtte [2]. The purpose of the investigation was to study the complex flow over the three-dimensional backward-facing step and to identify the noise-generation mechanisms for the given configuration.

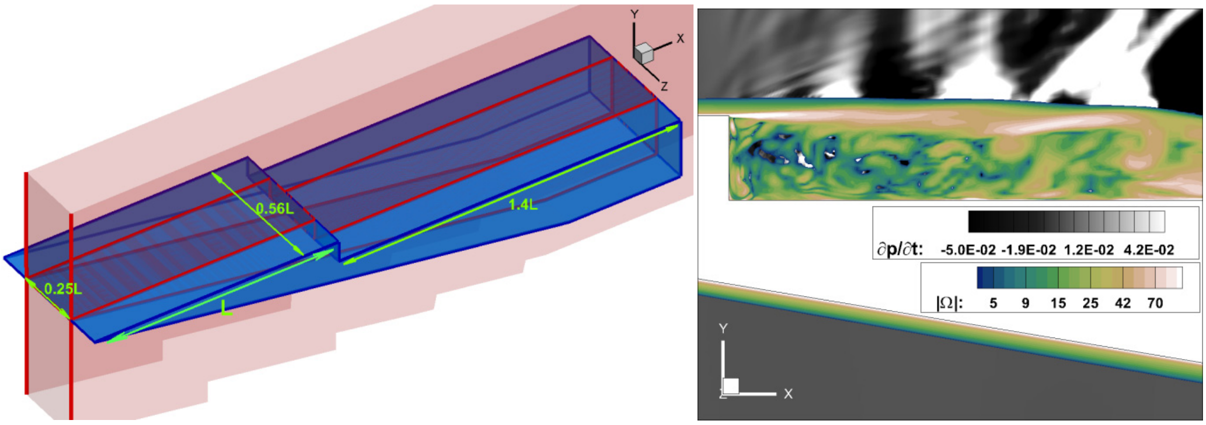

Fig. 1: Model of wedge with third-dimensional backward-facing step (left) and instanteneous flow field in the central 2D section downstream the step (right)

As seen from the instantaneous flow field on Fig. 1 (right), the numerical transition of the mixing layer stalled from the step edge to three-dimensional regime is delayed. It results in the significant downstream shift of the bottom-wall reattachment point at the distance about one step height with respect to the corresponding experimental value, while in general the numerical and experimental results were quite similar. It may be explained by the following reasons. First, it could happen due to inadequate grid resolution in the spanwise direction (Z) in the centre part of the model, which is about $\delta/3$ (δ – boundary layer thickness over the wall upstream the step). Second, it can be explained by the use of hybrid numerical scheme for inviscid fluxes. The point is that the pure central-difference scheme, when it is used on unstructured meshes, can cause an unstable behavior of the solution. To stabilize it, a small scheme dissipation in the LES zone is to be added. The problem is complicated by the existence of supersonic zones and shocks in the vicinity of the mixing layer.

To resolve this issue, a series of investigations was carried out for a simplified configuration presenting a cutout part of basic configuration of width 0.25L with the periodicity boundary conditions in the spanwise direction (see the area marked in red on Fig. 1 , left). The mesh resolution in all directions remained the same.

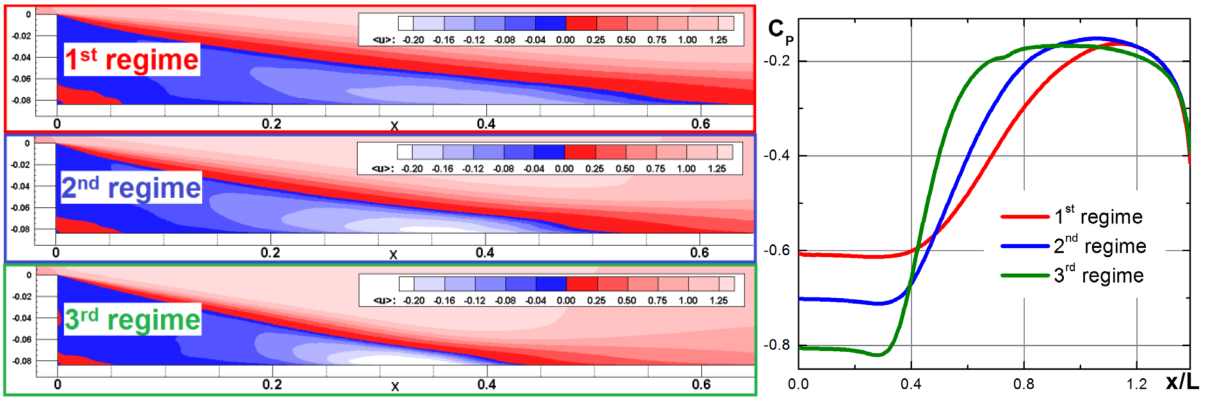

Fig. 2: Averaged streamwise velocity (left) and pressure coefficient on the bottom wall (right) downstream the backward-facing step from 3 obtained unsteady simulations

The IDDES computations were carried out in 3 regimes: using the upwind numerical scheme (1st regime); using the hybrid numerical scheme for inviscid fluxes (2nd regime); using the hybrid numerical scheme for inviscid fluxes and the modified definition of mesh length scale in the area downstream the backward step (3rd regime). The main detail of modification was the artificial reduction of mesh step Δz in the formula for $\Delta = \max\left(\Delta x, \Delta y, \Delta z\right)$. As a result, the numerical transition of the mixing layer to the unsteady 3D mode occurred earlier and noticeably changed the averaged flow field and pressure coefficient (see Fig. 2). Further, this technique was successfully applied to the fully 3D configuration.

All the above considerations are more or less related to the problem of “grey area” in the hybrid RANS/LES methods. The specificity is that we try to solve this problem on unstructured meshes.

REFERENCES

[1] Shur M.L., Spalart P.R., Strelets M.Kh., Travin A.K. A hybrid RANS-LES approach with delayed-DES and wall-modeled LES capabilities. *Int. Journal of Heat and Fluid Flow*, Vol. **29(6)**, pp. 1638-1649, 2008.

[2] Abalakin I.V., P.A. Bakhvalov, A.V. Gorobets, A.P. Duben, T.K. Kozubskaya, Parallel research code NOISEtte for large-scale CFD and CAA simulations. *Vychislitelnye Metody i Programmirovanie* 13: 110-125, 2012 (in Russian).