AR2 Test case
3D Transonic Channel with Swept Bump
HiOCFD4
FORTH - Heraklion

Florent Renac, Vincent Couaillier

ONERA, CFD department

ONERA
THE FRENCH AEROSPACE LAB

return on innovation

Specification

General description

- Objective : analyze HO methods capacity for a 3D shock wave / boundary layer
- Interactions taking place with the boundary layers of the four walls can lead to the formation of several separations
- Experiments executed at Onera by Délery team provide a thorough description of the flow field and detailed experimental data are available
- Reference FV computations (to be completed by very fine mesh computations) with RANS/2 eq. turbulence models have been performed
- The target quantities of interest are the static pressure distribution on the walls, the turbulent kinetic energy profiles and the mean stream-wise velocity profiles in longitudinal planes.

Governing equations

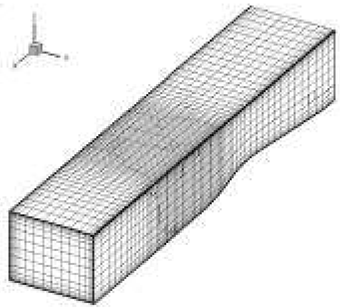
- 3D Reynolds-averaged Navier-Stokes system
- Dynamic viscosity is prescribed with the Sutherland law
- Turbulence model is left up to the participants; recommended suggestions are :

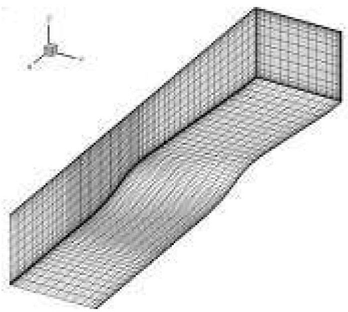
1) Spalart Allmaras model
2) Wilcox k-omega model
3) k-omega SST model

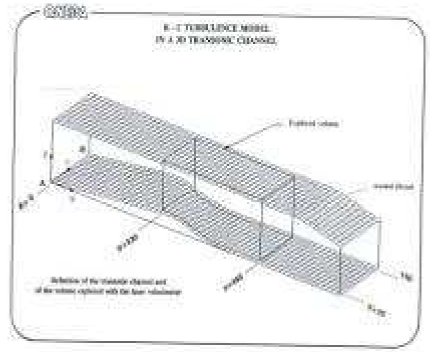
Test case description 1/3

Geometry

- It consists of a converging-diverging section with three flat faces and the fourth face (lower wall) bearing a swept bump







Meshes

- Participants may use their own series of grids, but straight and high order hexahedra meshes will be provided by Onera.
- P2 meshes have been generated

Experimental data :

Experimental data will be available in order to verify the consistency of the simulations with physics but not as reference for converged solutions in terms of DoF's (modelling of turbulence models will necessary introduce a gap).

These data are :

- Pressure distribution on the 4 walls : Pdist
- Turbulent kinetic energy profiles in several stations of longitudinal planes : Kprof
- Mean stream-wise velocity profiles in several stations of longitudinal planes : Vprof
- RMS on fluctuation velocity: Reynolds stress components

Test case description 2/3

Computation outputs

Extract the following quantities:
- Pressure distribution on the 4 walls : Pdist
- Mean stream-wise velocity profiles in several stations of longitudinal planes : Vprof
- Optional :
    - Turbulent kinetic energy profiles in several stations of longitudinal planes : Kp
    - RMS profiles : RMSprof

1- Describe the numerical scheme together with the technique(s) used for DoF's analysis

2- Perform a convergence study of Pdist, Vprof with at least one of the following techniques :
- H refinement using uniform refinement or local refinement with local error estimator
- P refinement using uniform refinement or local refinement with local error estimator
- H/P refinement using uniform refinement or local refinement with local error estimator

3- Optionally if the turbulence model used allows this : perform a convergence study of Kprof

Test case description 3/3

Flow conditions

- Fully turbulent transonic flow
- RANS equation (S.A. Model, $K\omega$ model, …)
- Reynolds number (throat height, stagnation conditions) :

$$Re\,y_i = \frac{\rho_i c_i H_t}{\mu(T_i)} = 1.13 \times 10^7$$

Boundary conditions

- Fully turbulent transonic flow
- RANS with SA model

$$\frac{P_{s,outlet}}{P_{i,inlet}} = 0.58$$

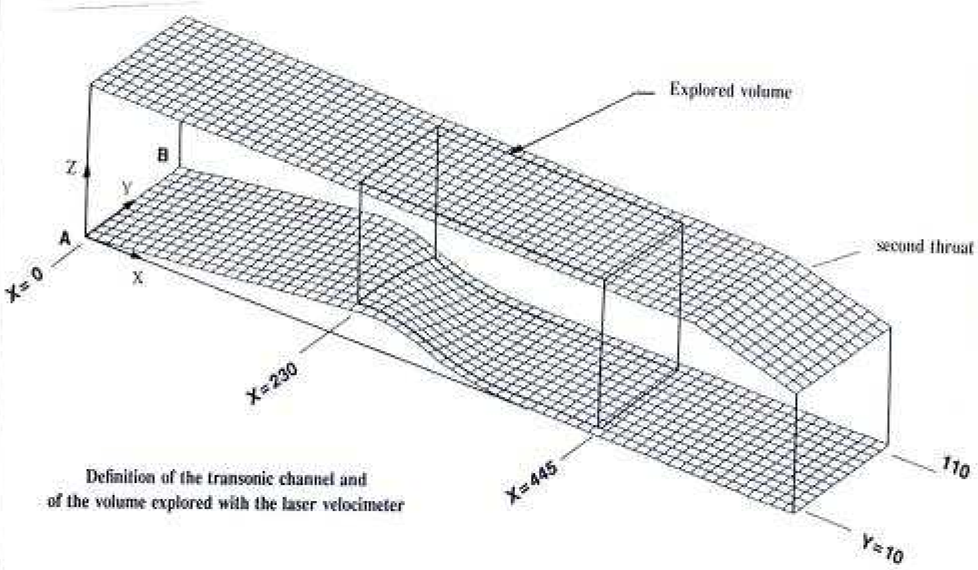
-

Reference data (Délery & Marvin 1986 ; Benay & Pot 1985)

- Wall pressure measurements
- LDV velocity measurements (mean components, RMS of fluctuations)
- Turbulent shear stresses

CALCUL CFD DANS UN CANAL 3D TRANSSONIQUE (1)
VALIDATION DE MODÈLES DE TURBULENCE

K – ε TURBULENCE MODEL
IN A 3D TRANSONIC CHANNEL



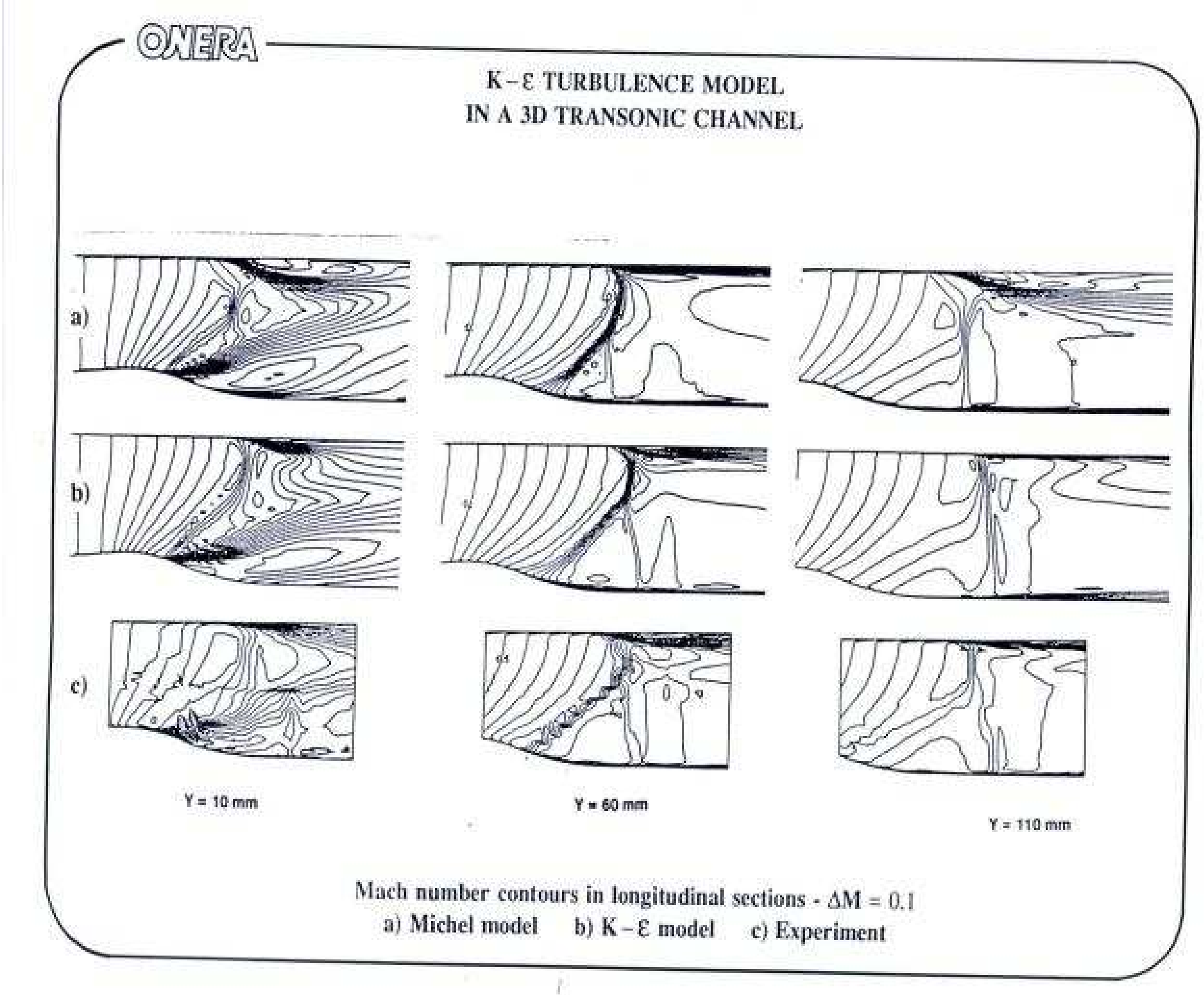
Definition of the transonic channel and of the volume explored with the laser velocimeter

K – ε TURBULENCE MODEL IN A 3D CHANNEL

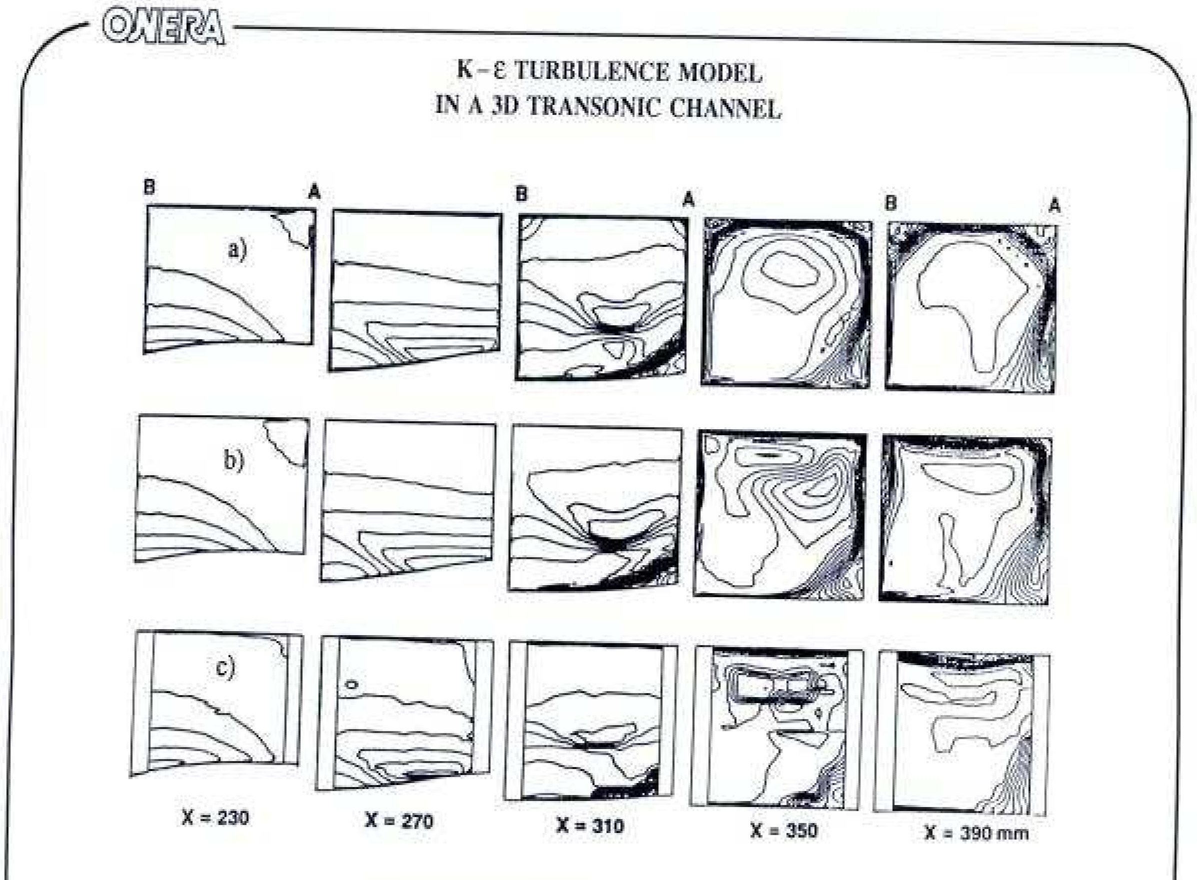
COMPUTATION

- ☐ **Mesh :** 153 x 65 x 61 = 606,45 **points**
- ☐ **Initial conditions : 1-D inviscid flow with a shock located in the divergence part of the channel**
- ☐ **Number of iterations : 5000 with the Michel model and 10000 with the K – ε model**
- ☐ **Multigrid used with three grids (two coarse grids)**
- ☐ **CPU time :** 14μs / it /pt with the Michel model
  20μs / it /pt with the K – ε model

CALCUL CFD DANS UN CANAL 3D TRANSSONIQUE (2)
VALIDATION DE MODÈLES DE TURBULENCE

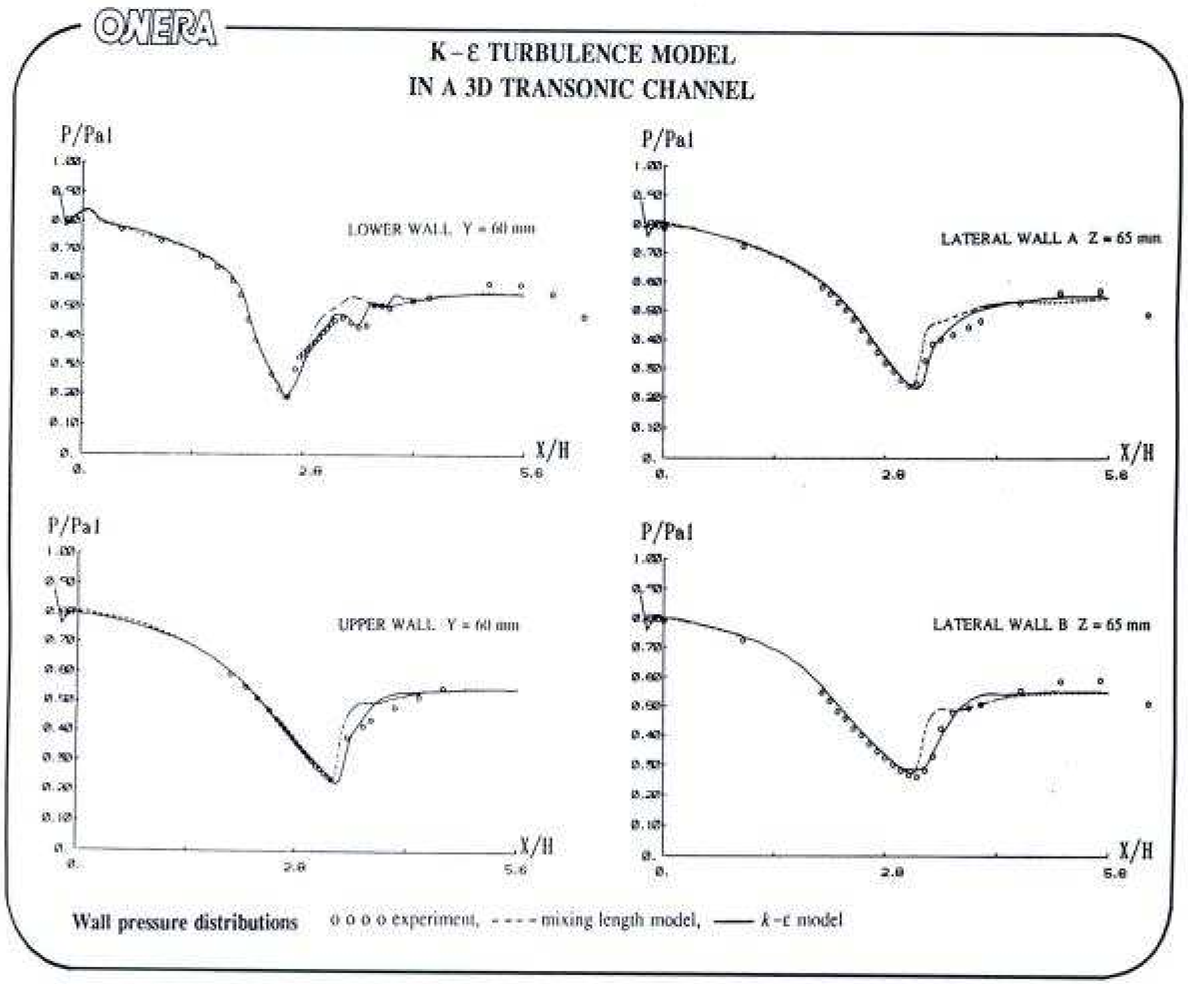


CALCUL CFD DANS UN CANAL 3D TRANSSONIQUE (3)
VALIDATION DE MODÈLES DE TURBULENCE

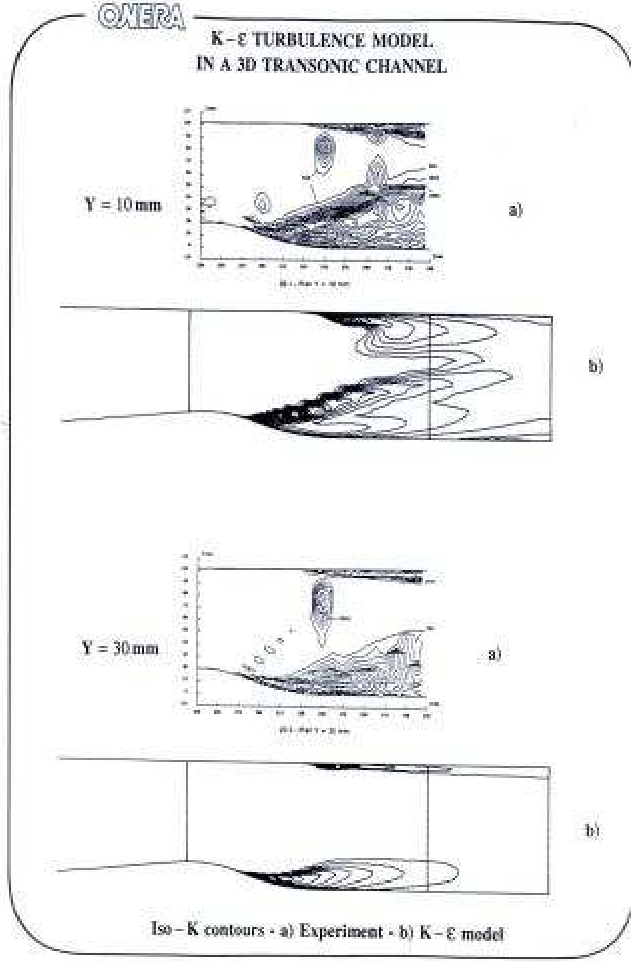


Mach number contours in transverse sections - ΔM = 0.1
a) Michel model b) K – ε model c) Experiment

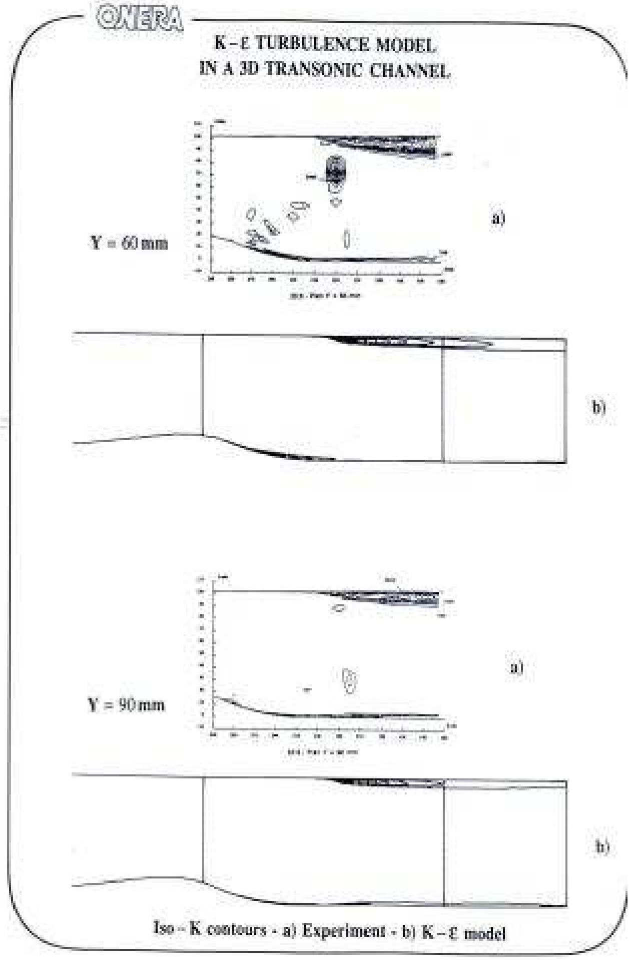
CALCUL CFD DANS UN CANAL 3D TRANSSONIQUE (4)
VALIDATION DE MODÈLES DE TURBULENCE



CALCUL CFD DANS UN CANAL 3D TRANSSONIQUE (5)
VALIDATION DE MODÈLES DE TURBULENCE

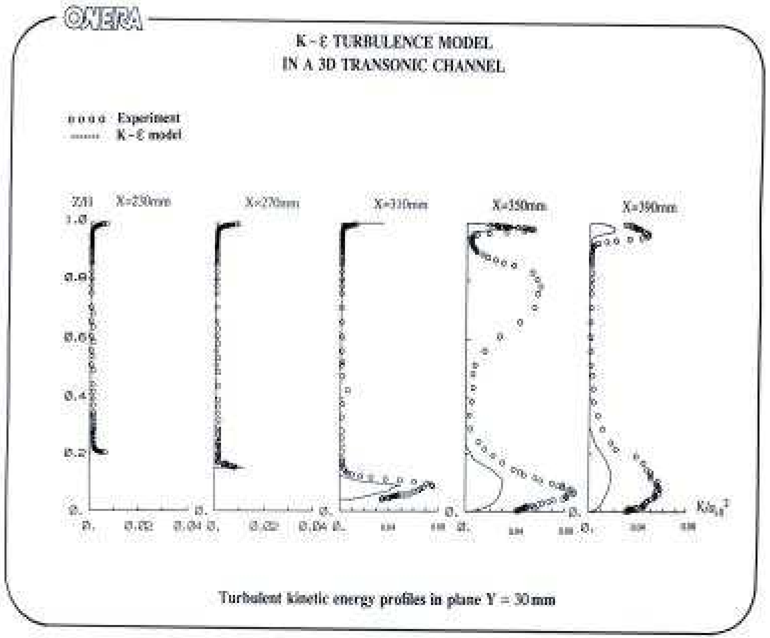


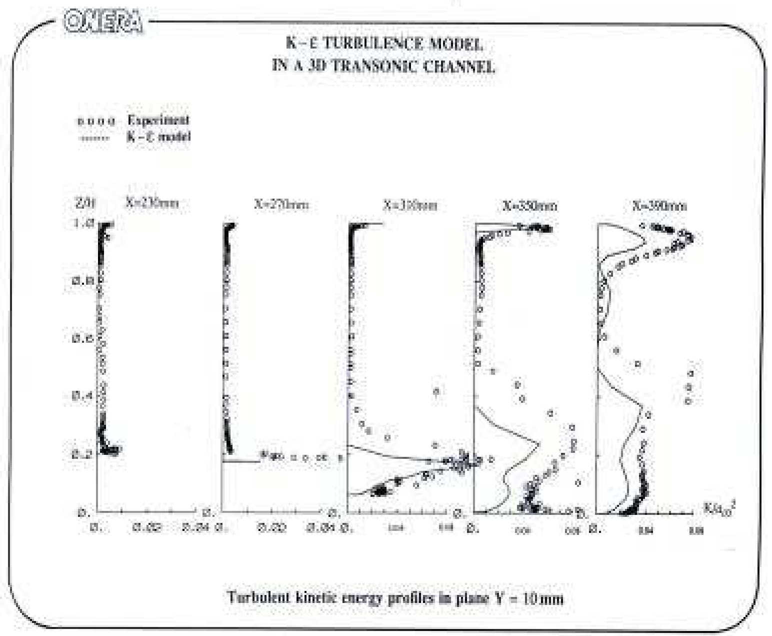
Iso – K contours - a) Experiment - b) K – ε model



Iso – K contours - a) Experiment - b) K – ε model

CALCUL CFD DANS UN CANAL 3D TRANSSONIQUE (6)
VALIDATION DE MODÈLES DE TURBULENCE

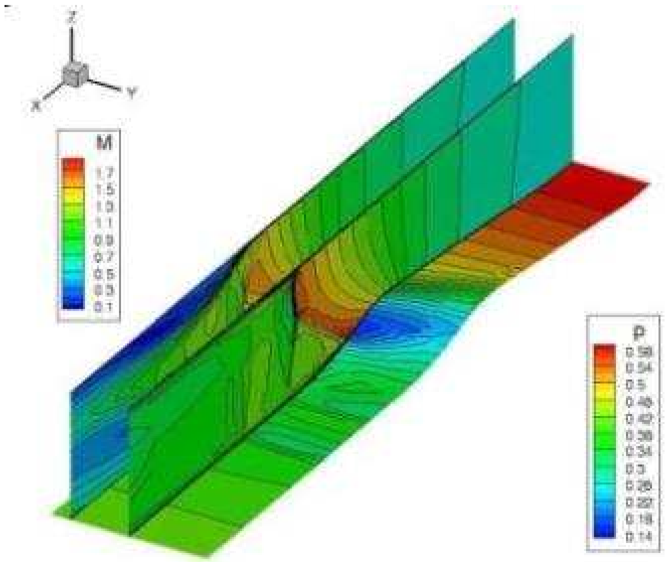




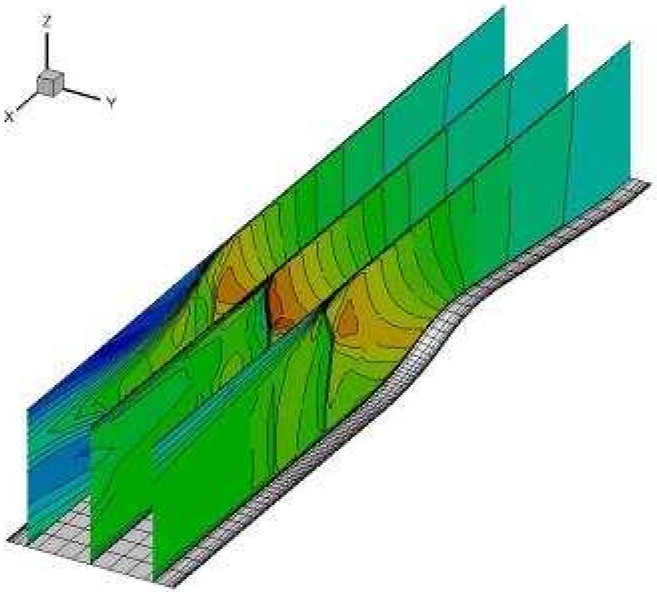
Computations with Aghora solver

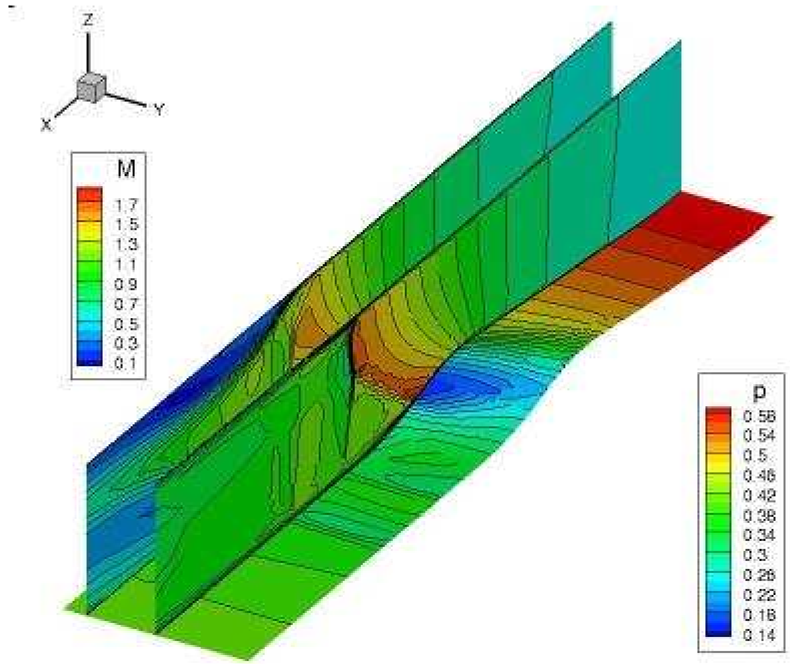
Aghora solver : Discontinuous Galerkin method

- Polynomial degree $1 \leq p \leq 3$
- SA model
- Convection : LLF flux
- Diffusion : BR2
- Shock capturing technique based on entropy production
- Quadratic meshes 72,950 nodes with flow separation ($y^+ = 4$ before separation for p=2) and 540,000 nodes
- Partition : First results into 48 cores, new one into into 2400 cores with OCCIGEN (French GENCI project 2015-2016)
- Robust shock capturing method (Guermond at al.)
- Bacward Euler scheme
- Damped inexact Newtion method (GMRES with ILU(0) preconditionning)
- Jacobian-free matrix-vector product
- CFL evolution from pseudo-transient continuation technique $CFL \leq 10^{10}$

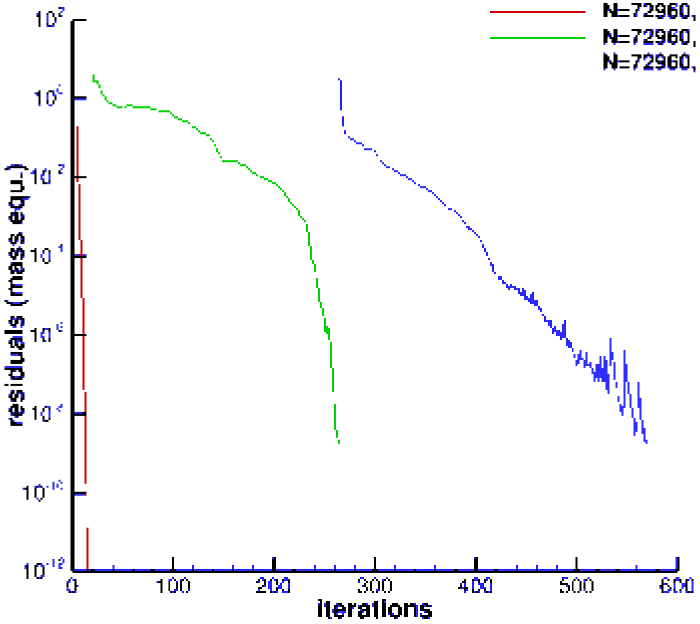


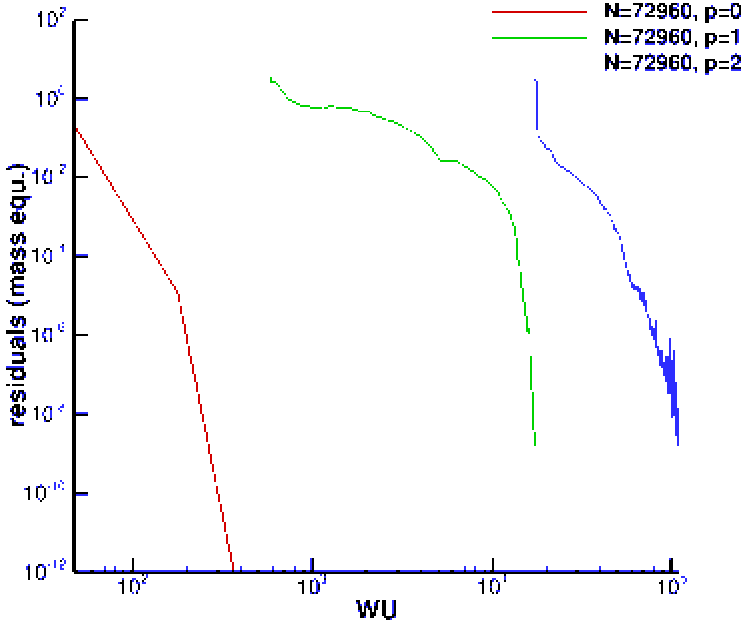
Aghora – 3D view



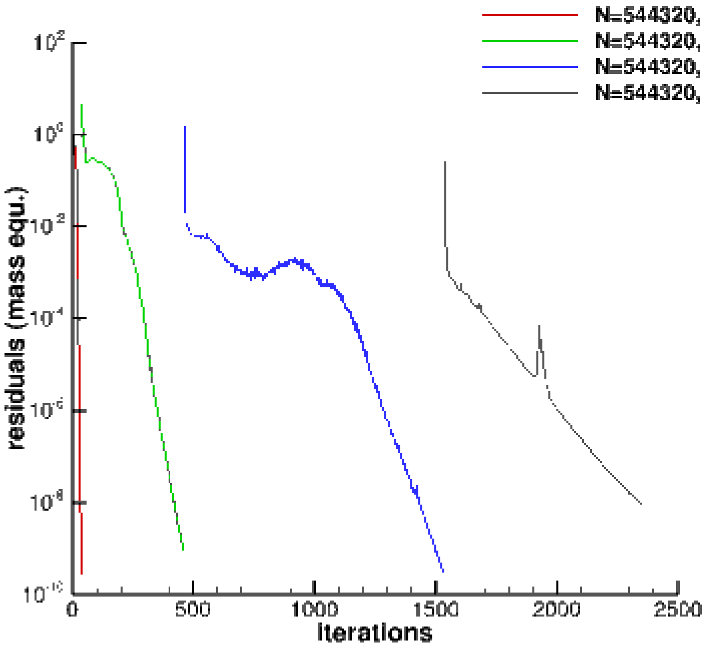


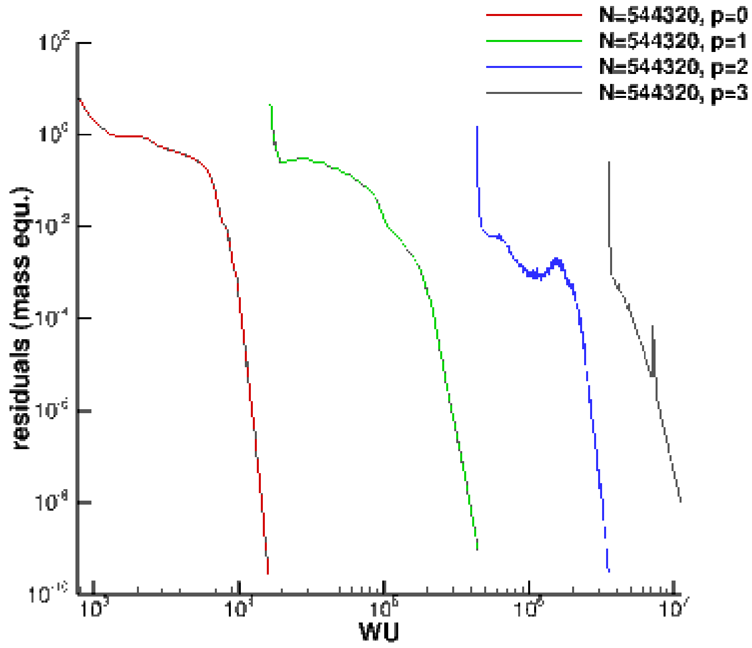
Aghora : Iterative convergence – Mesh 73K





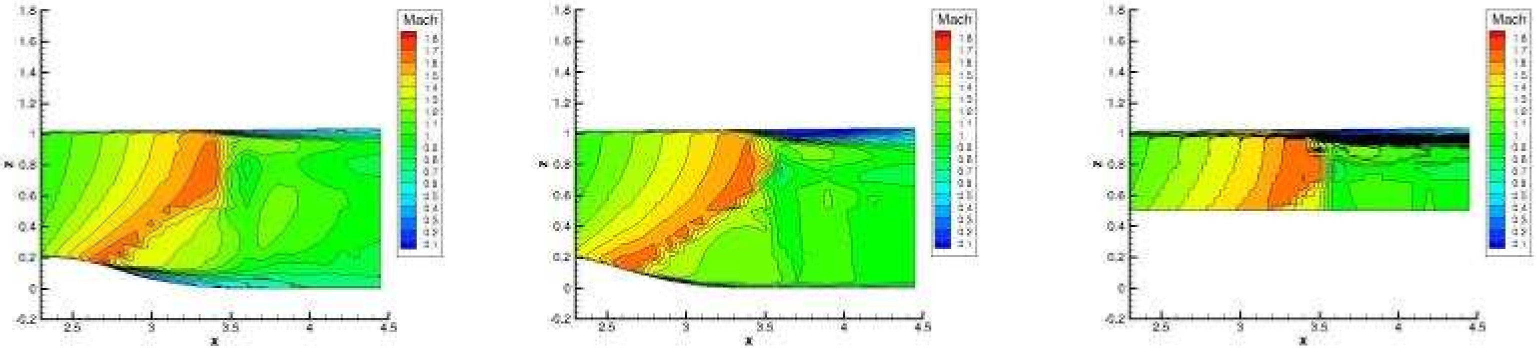
Aghora : Iterative convergence – Mesh 540K



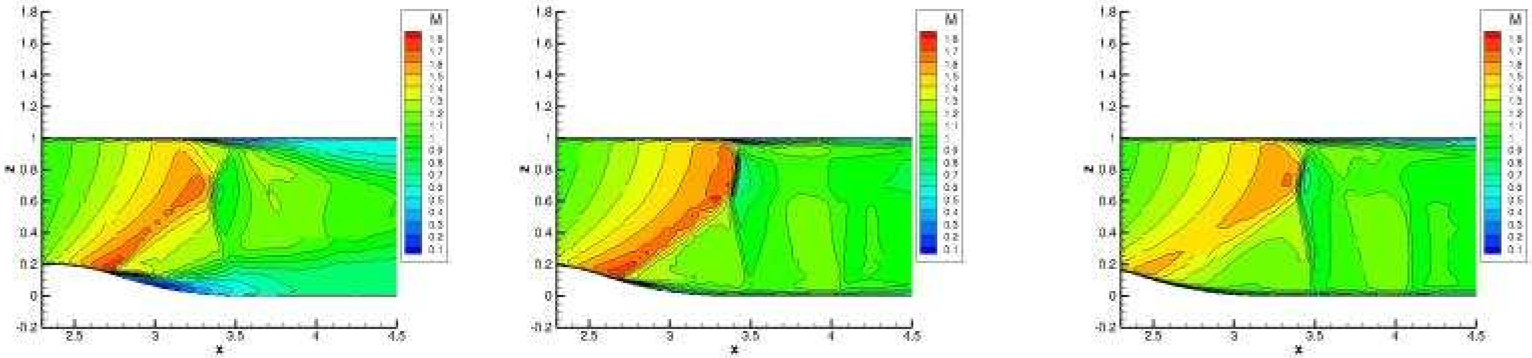


Aghora : Mach contours – Mesh 73K – P=3

Y=30 Y=60 Y=90

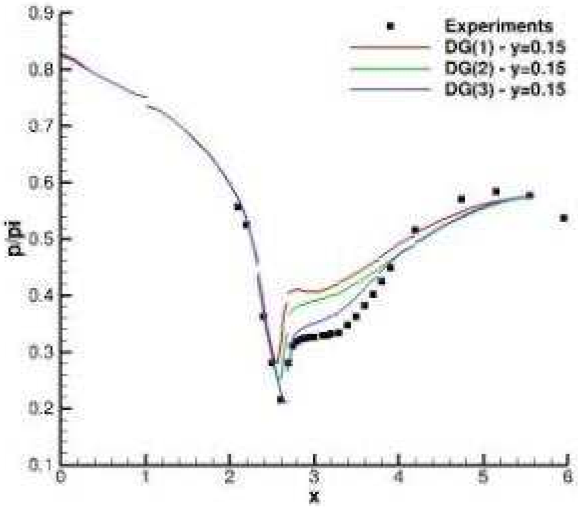


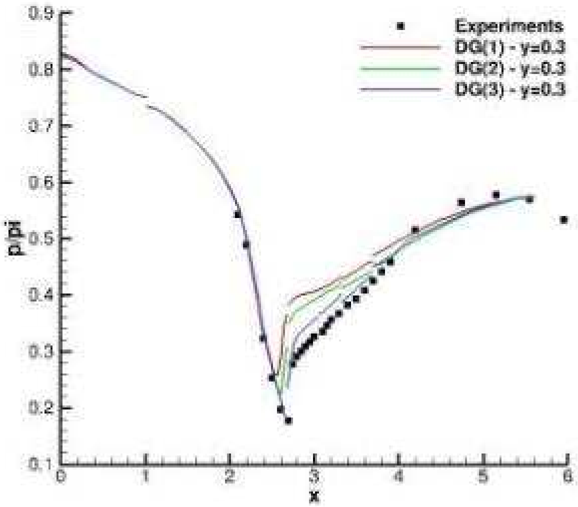
Experimental result (ONERA)

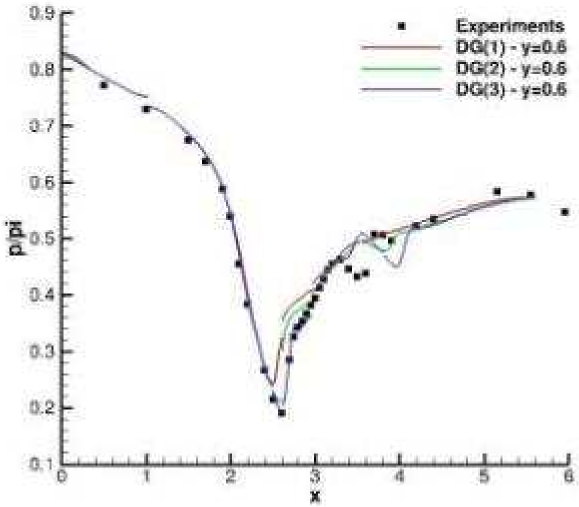


Aghora results with S.A. model with P3 approximation

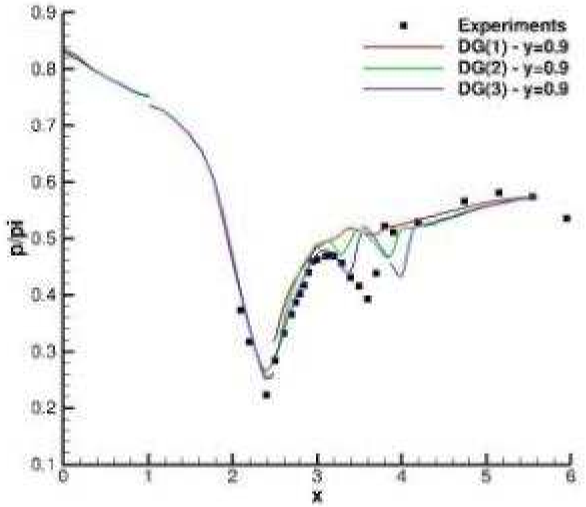
Aghora : Pressure on lower wall – Mesh 73K

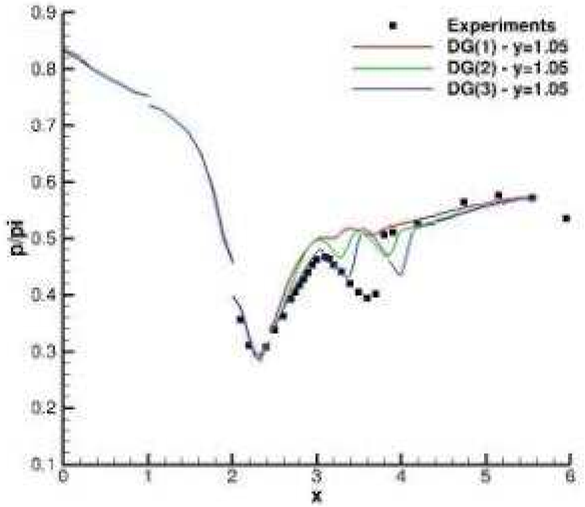






Aghora : Pressure on lower wall – Mesh 73K

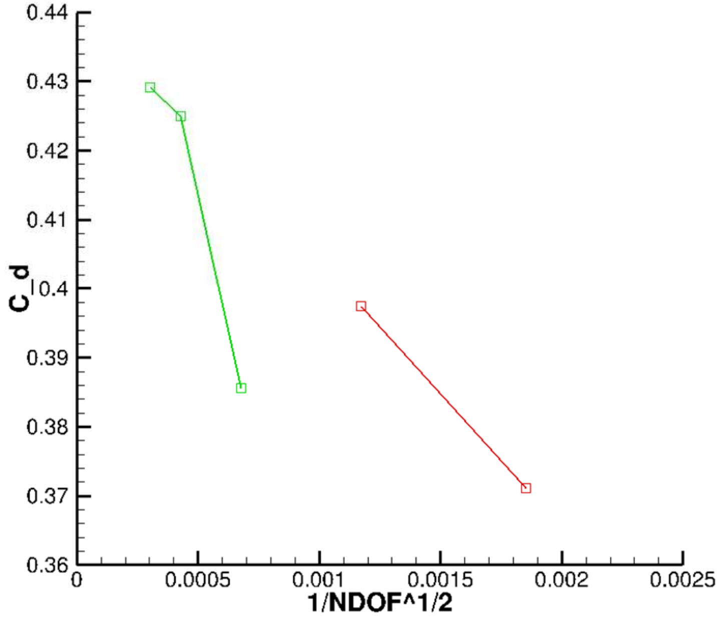


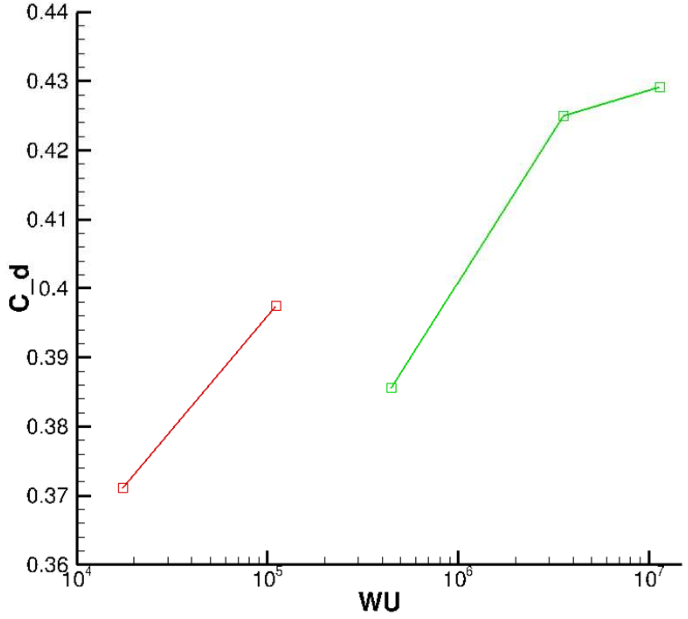


ONERA
THE FRENCH AEROSPACE LAB

Aghora : Drag coefficient

Red : Coarse mesh 77K   -   Green : medium mesh 540 K



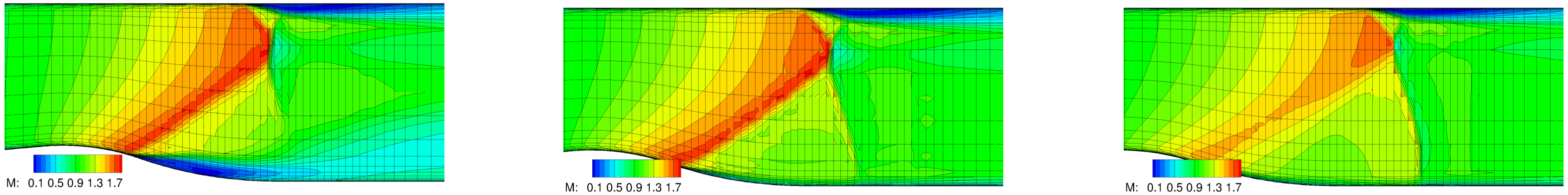


3D swept bump : on-going work for ONERA

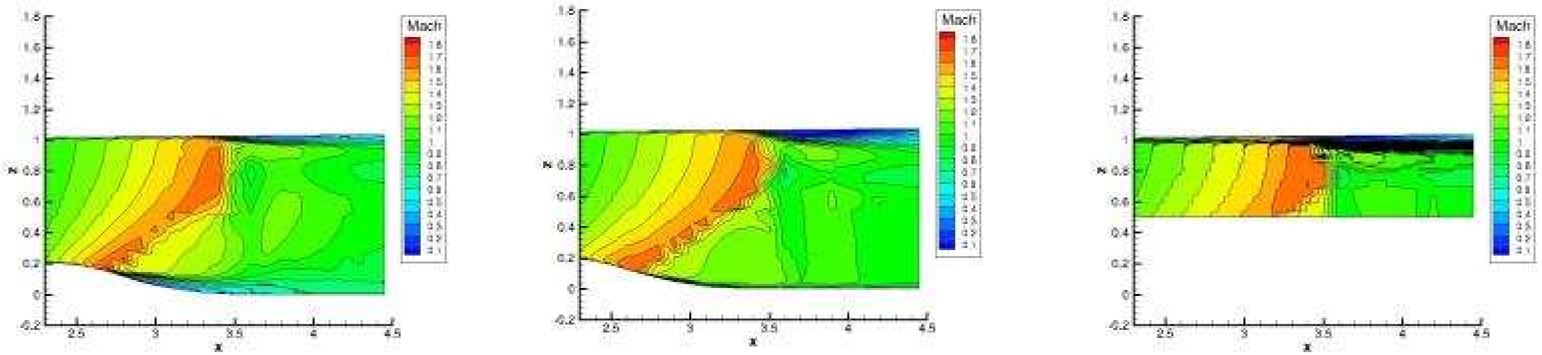
- Apply h/p refinement to converge in terms DOFs for drag coefficient
- Unsteady computations with ZDES (Zonal DES)
- Modify outlet boundary condition (second throat)

Mach number – Comparison Aghora/Migale/experiments
Coarse mesh 73K

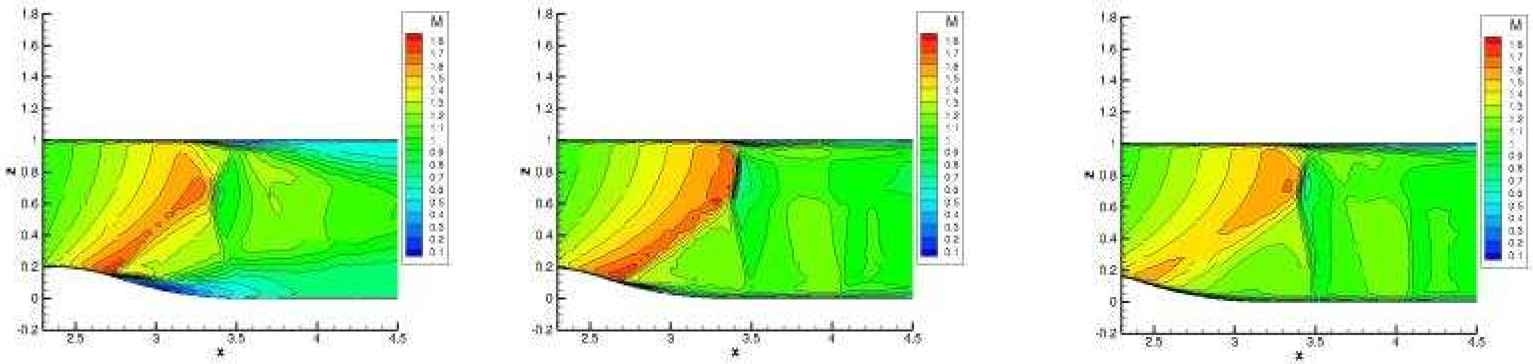
Y=30 *Y=60* *Y=90*



MIGALE results with Low Reynolds kω model with P2 approximation (U. Bergamo)



Experimental result (ONERA)

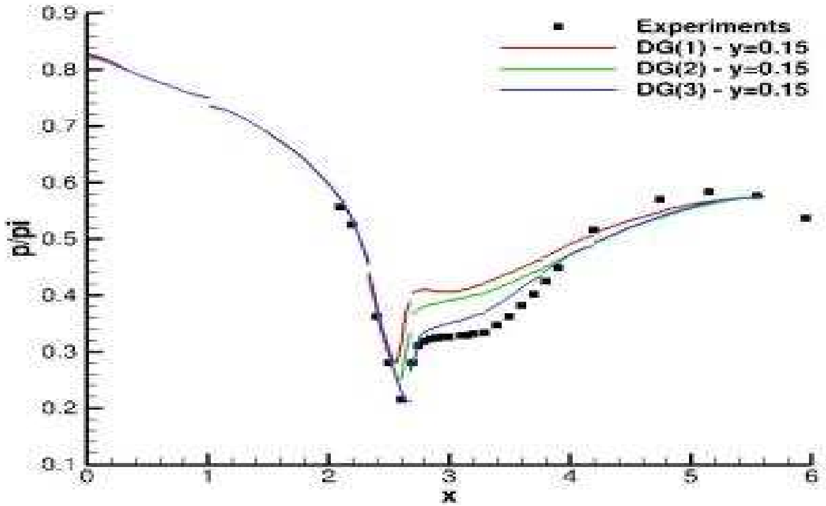


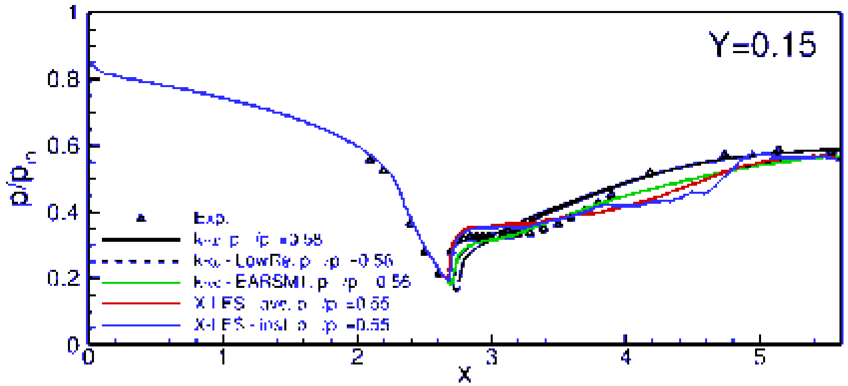
Aghora results with S.A. model with P3 approximation (ONERA)

ONERA

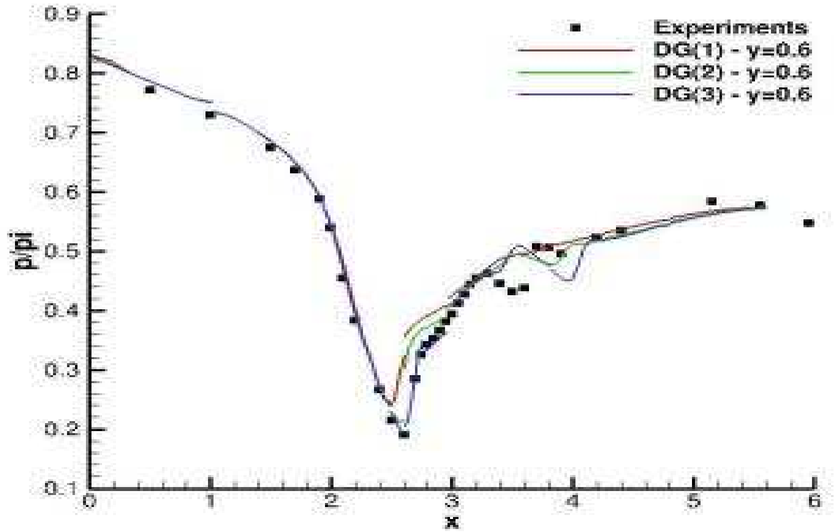
THE FRENCH AEROSPACE LAB

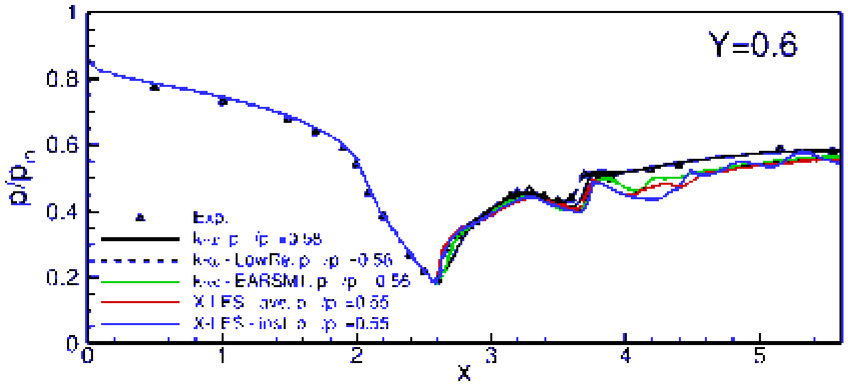
Pressure – Comparison Aghora/Migale/experiment
Coarse mesh 73K – Y=15



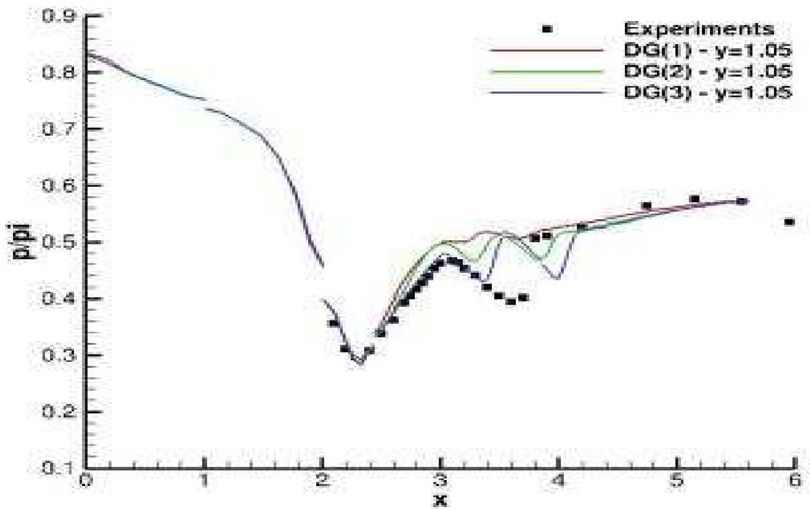


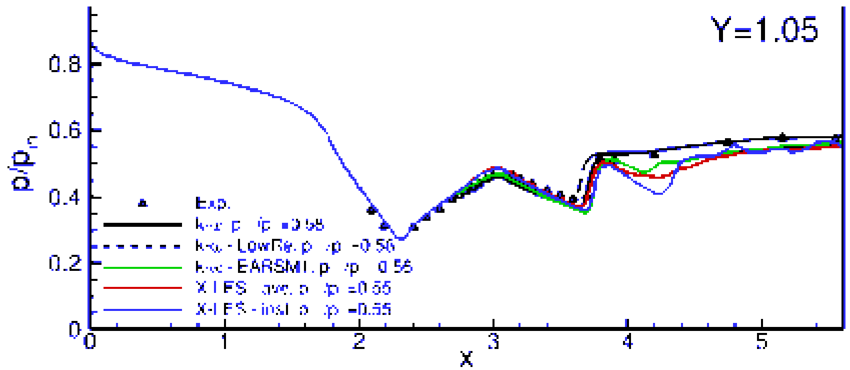
Pressure – Comparison Aghora/Migale/experiment
Coarse mesh 73K – Y=60





Pressure – Comparison Aghora/Migale/experiment
Coarse mesh 73K – Y=105





3D swept bump : Next steps

- Define a convergence criterium (friction forces on walls)

- Modify outlet boundary condition (second throat)