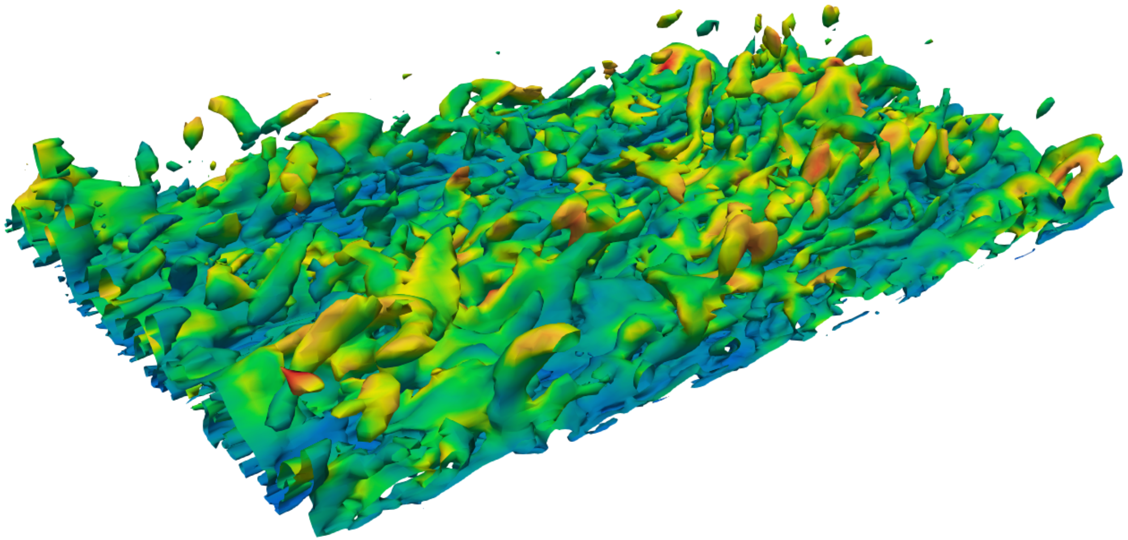
Turbulence models assessment using finite-volume and high-order methods for aeronautical applications

Bernat Font*, Fabio Naddei, Oriol Lehmkuhl



Barcelona Supercomputing Center
Centro Nacional de Supercomputación
ONERA
THE FRENCH AEROSPACE LAB
NextSim
Next generation of industrial aerodynamic simulation code
EuroHPC Joint Undertaking

Discretization methods, turbulence models, and mesh types, affect convergence

Spatial discretization methods:

- Finite Difference
- Finite Volume
- Finite Element
- Discontinuous Galerkin

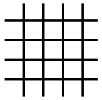
....

LES turbulence closures:

- Smagorinsky
- Dynamic Smagorinsky
- WALE (wall-adapting local eddy-viscosity)
- Vreman

...

Mesh



- There is no obvious choice to match discretization method and turbulence model.
- Turbulence models usually are case- (and mesh-) dependent too.

$$\tau_{ij}^a = -2\nu_{\mathrm{SFS}}\bar{S}_{ij}$$

$$\phi = \langle\phi\rangle + \phi'$$

The integral length scale approximation (ILSA) model

The filter width $\triangle$ depends on an estimated integral scale, and not on the local cell size.
The integral scale is calculated with temporal-averaged quantities.

$$\nu_{\mathrm{SFS}} = (C_k L_{\mathrm{est}})^2 \left|\bar{S}\right| \qquad \triangle = C_\triangle L_{\mathrm{est}} \qquad L_{\mathrm{est}} = \frac{\langle\mathcal{K}_{\mathrm{res}}\rangle^{3/2}}{\langle\varepsilon_{\mathrm{tot}}\rangle}$$

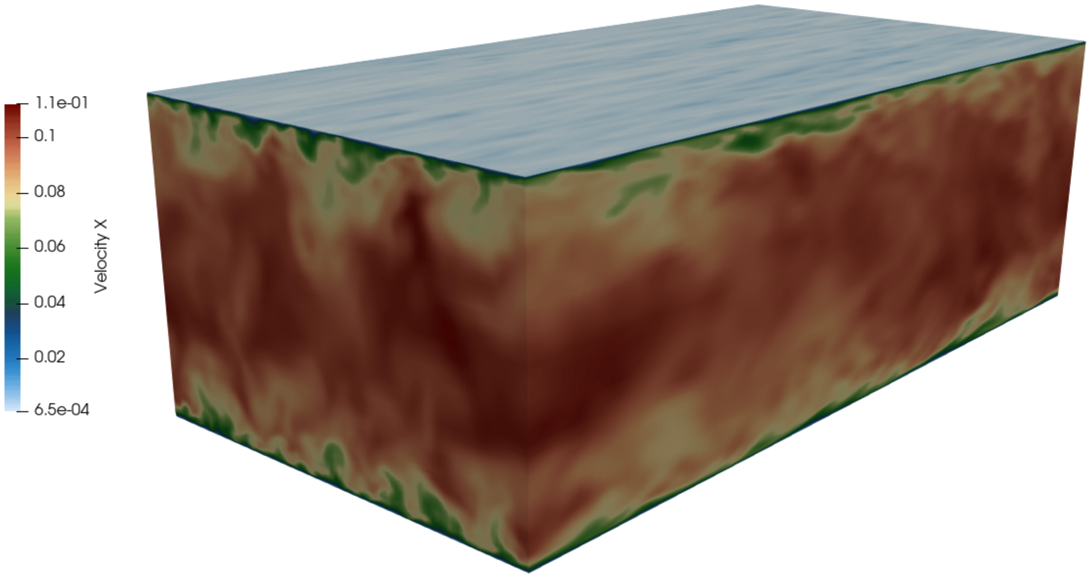
C_k is determined via s_τ, ILSA's single parameter which encapsulates the amount of turbulent kinetic energy aimed to be modelled.

$$s_\tau = \left[\frac{\langle\tau_{ij}^a\tau_{ij}^a\rangle}{\langle(\tau_{mn}^a + R_{mn}^a)(\tau_{mn}^a + R_{mn}^a)\rangle}\right]^{1/2} \qquad \mathcal{X}_1[1-(1/s_\tau)^2]C_k^4 - \mathcal{X}_2 C_k^2 + \mathcal{X}_3 = 0$$

- Because mesh independence, ILSA yields grid converged results for a range of s_τ .
- Suitable for h-adaptivity, and p-adaptivity in high-order methods.

1. A. Rouhi, and U. Piomelli, 2016. Phys. Rev. Fluids 1, 044401.
2. O. Lehmkuhl, U Piomelli, and G Houzeaux, 2019. Int. J. Heat and Fluid Flow.

Turbulent channel flow at $Re_\tau = 950$



Geometry: $6h \times 2h \times 3h$

$Ma = 0.08$

BC: Top & bottom walls: no-slip.
Others: periodic.

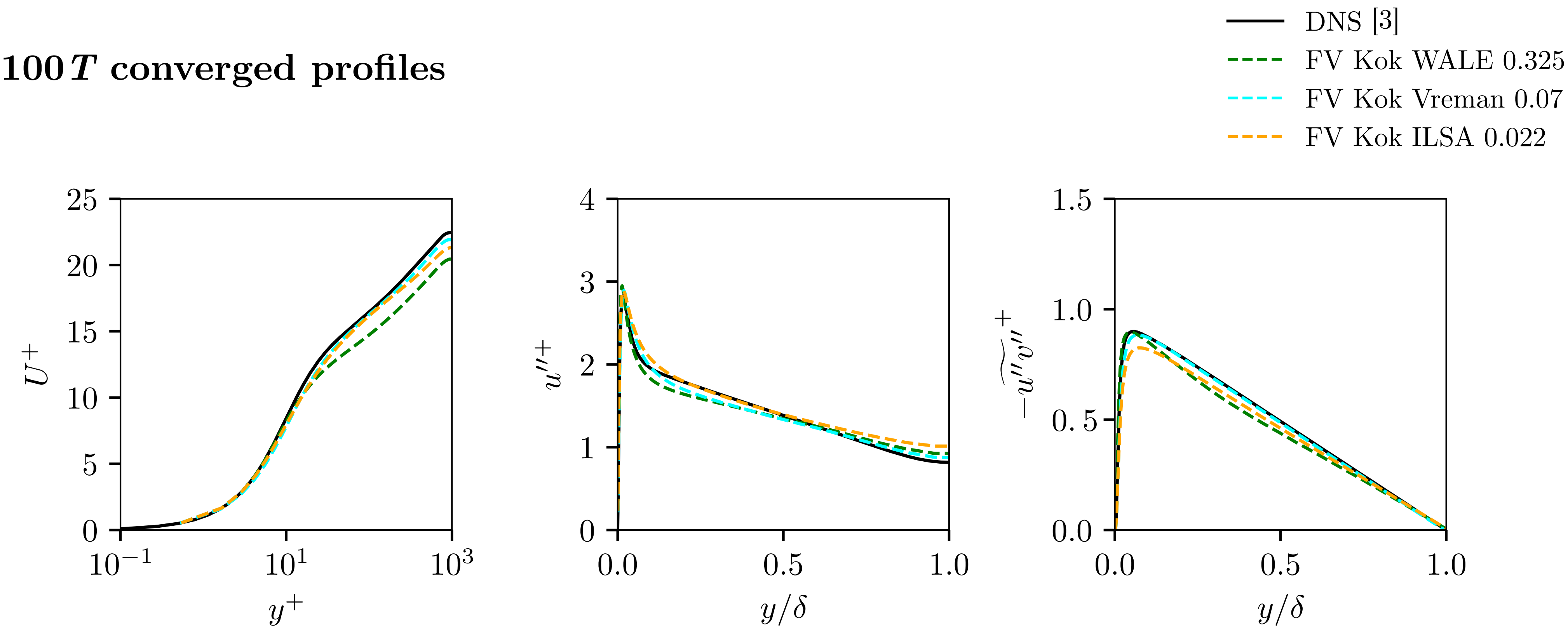
IC: Perturbed analytical solution.

Statistics averaged for $100\,T$.

Mesh: $96 \times 98 \times 96$
$y^+ = 1$

Numerical methods: **Finite volume**, Kok convective scheme, Runge-Kutta 4th order.

$100\,T$ converged profiles

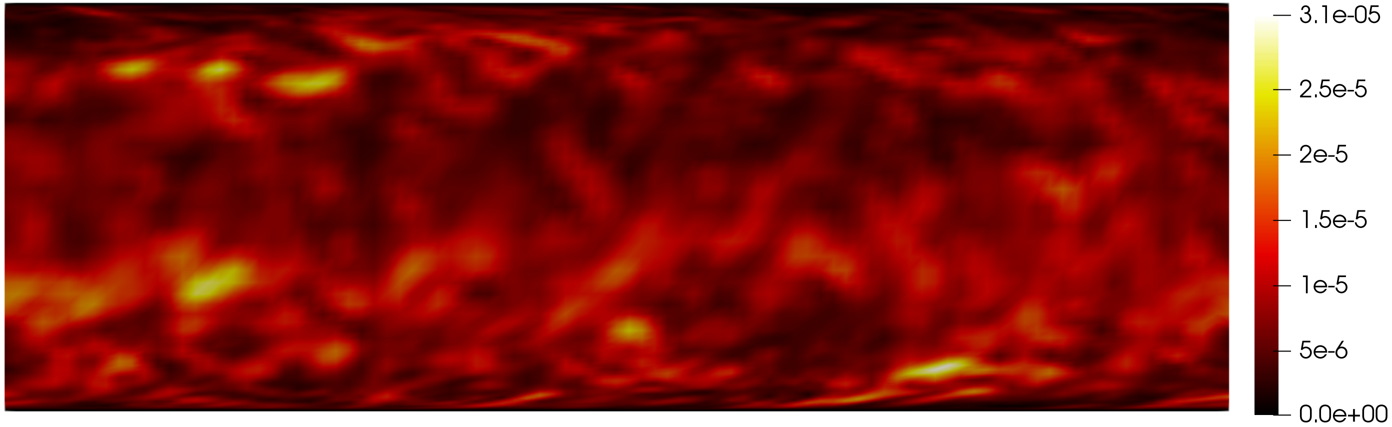


- All models have been tested with their default constants.
- ILSA and Vreman models yield similar results, while WALE displays a larger discrepancy.

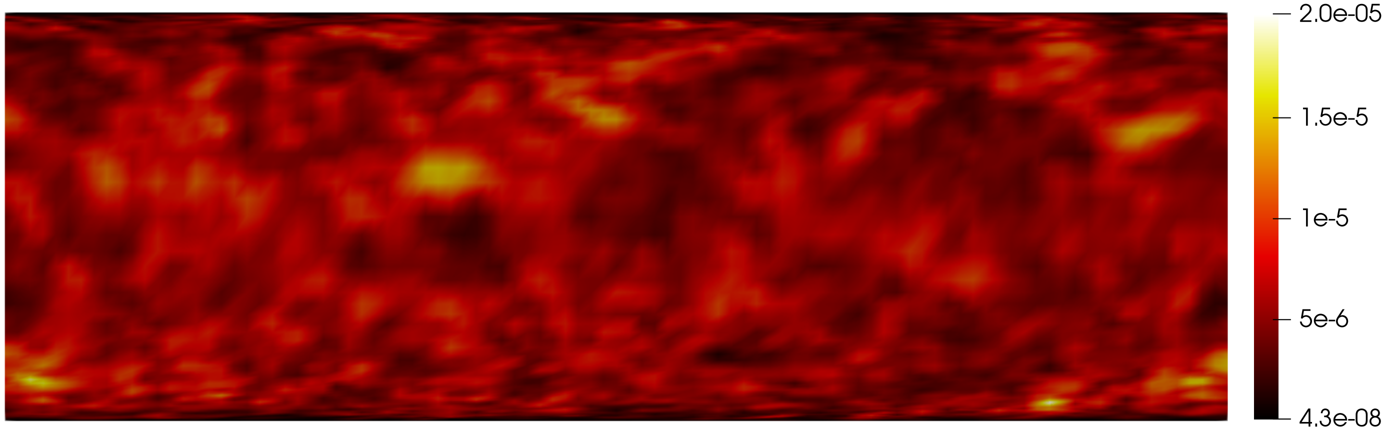
3. Hoyas & Jimenez, 2008. Phys. Fluids 20, 101511

The ILSA SGS viscosity follows physical structures despicte mesh anisotropy

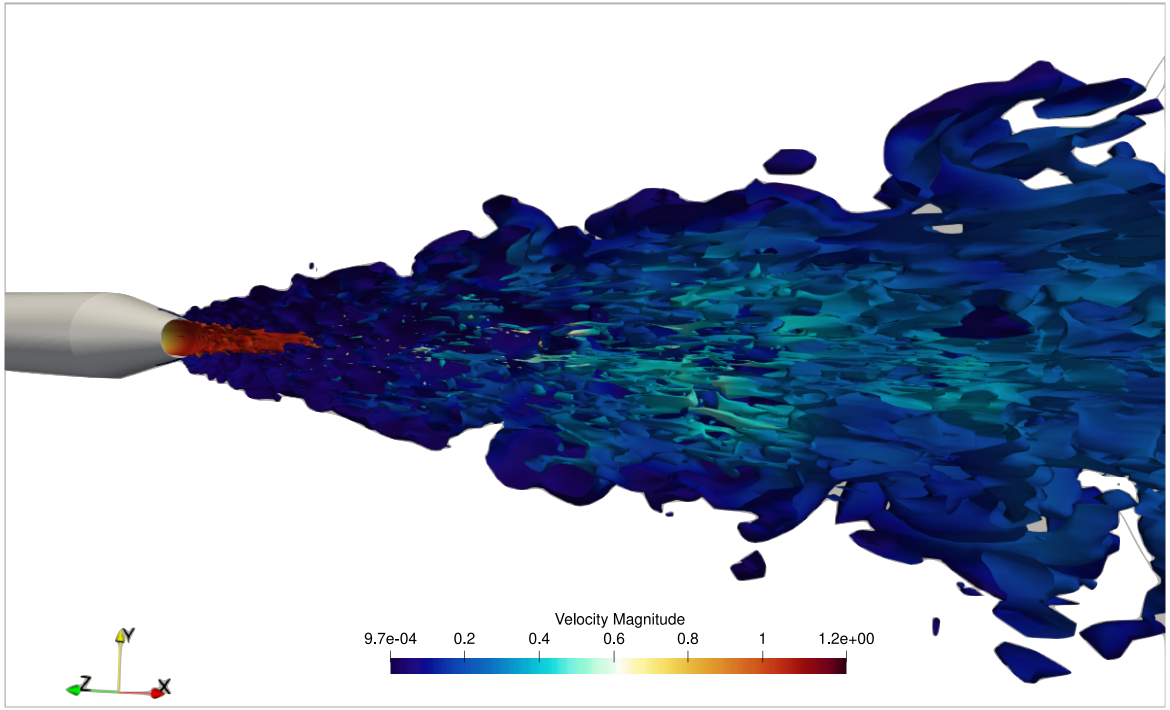
ILSA
$s_\tau = 0.022$



Vreman
$c \approx 2.5C_s^2 = 0.07$



Jet exhaust aerodynamics and noise (JEAN)* test case



Case kindly provided by Dr. S. Pezzano @ CERFACS

$Re = 10^6$

$Ma = 0.9$

IC: RANS-SA using FV + p staging

Mesh: 138M DOFs (2.1M Hex64)

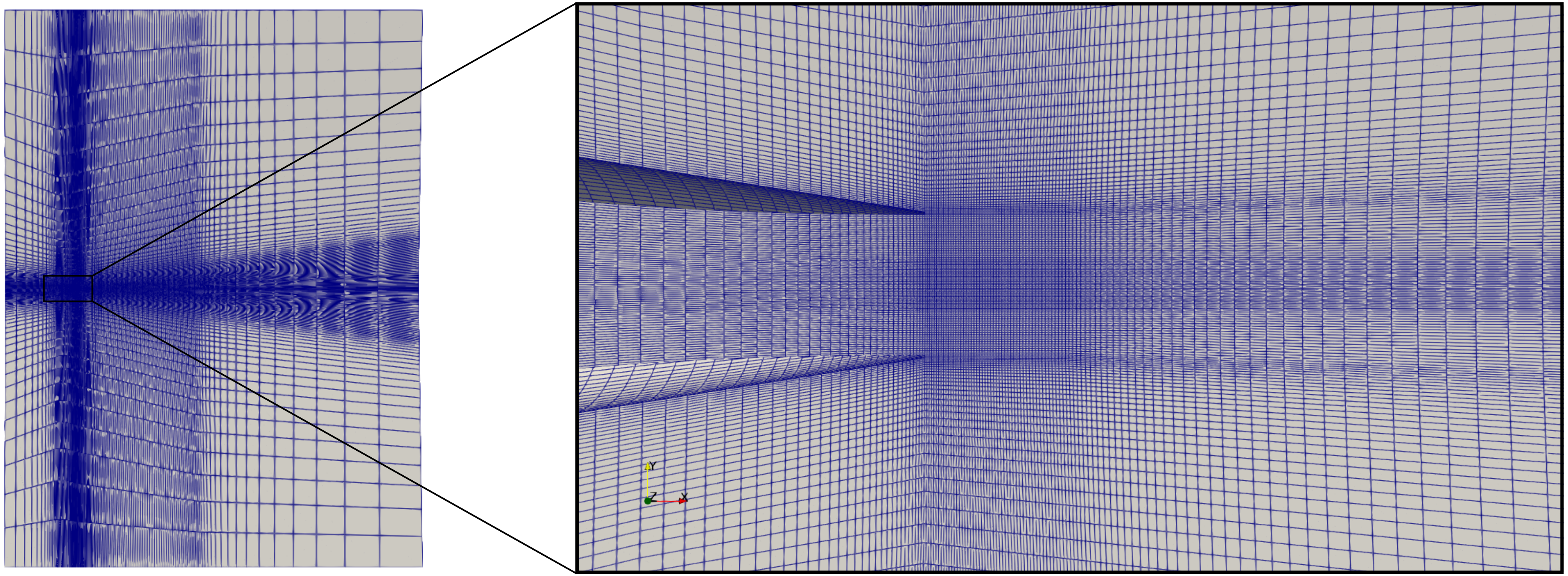
Numerical methods:

Discontinuous Galerkin spectral element method (**DGSEM**) with p=3

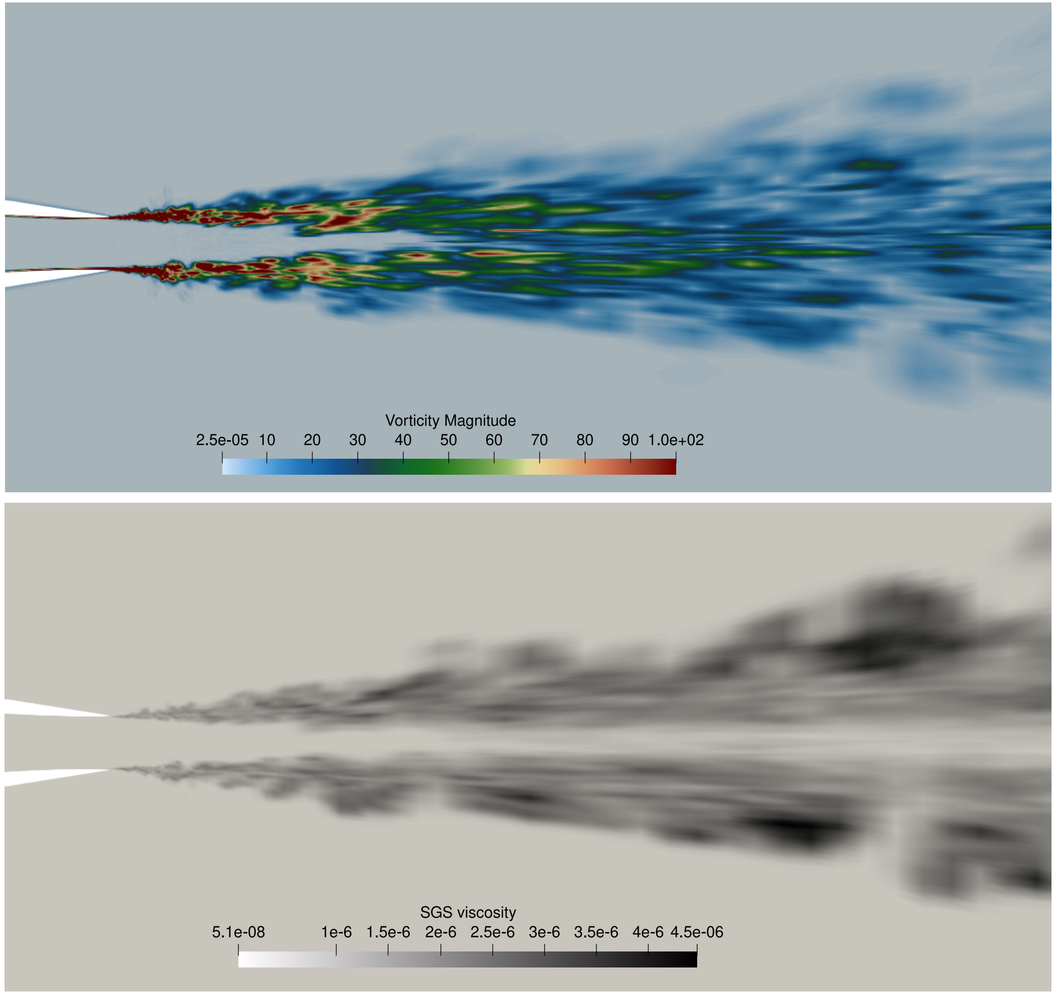
SLAU2 convective scheme
Runge-Kutta 3rd order time scheme

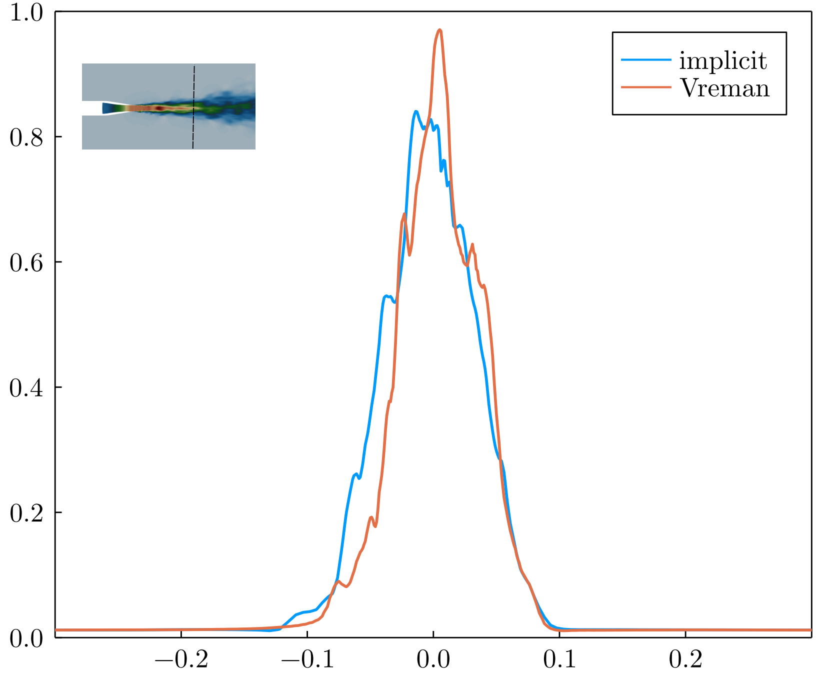
* https://cordis.europa.eu/project/id/G4RD-CT-2000-00313

A highly anisotropic mesh can be challenging for turbulence models



The Vreman model is desactivated in the center of the channel





Summary

Channel test case

- The WALE, Vreman, and ILSA LES models have been assessed.
- Vreman and ILSA achieved better results than WALE.
- The ILSA eddy viscosity follows the flow structures, contrary to the WALE model.

JEAN nozzle test case

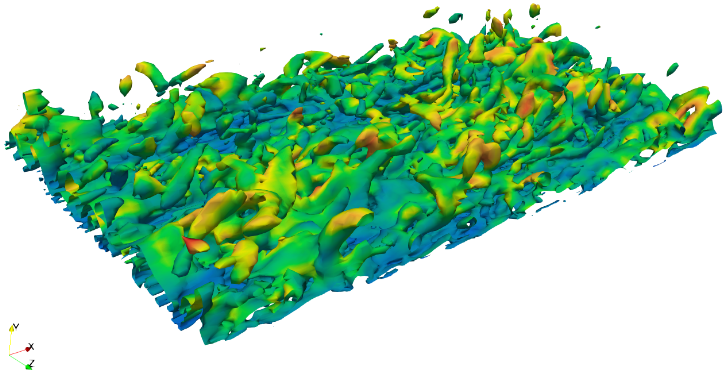
- The Vreman model has been compared with an implicit LES solution.
- The Vreman model gets desactivated at the centerline of the jet in the downstream region because of the highly anisotropic mesh.
- The jet profile appears more accelerated when using the Vreman model in comparison with iLES.

Future work

- Run the chan test case with the ILSA model.
- Run the JEAN test case with the ILSA model.

Turbulence models assessment using finite-volume and high-order methods for aeronautical applications

Bernat Font*, Fabio Naddei, Oriol Lehmkuhl



Thanks!

Download the presentation

@bernat_font

Barcelona Supercomputing Center
Centro Nacional de Supercomputación

ONERA
THE FRENCH AEROSPACE LAB

NextSim
Next generation of industrial aerodynamic simulation code

EuroHPC
Joint Undertaking