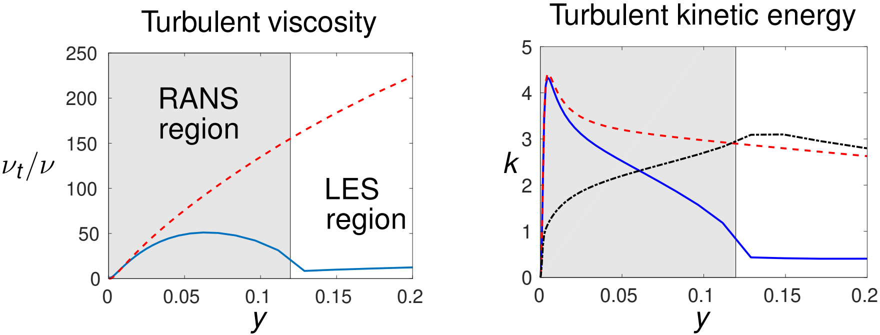
S-ZDES: Zonal Detached Eddy Simulation coupled with steady RANS in the wall region

Lars Davidson, www.tfd.chalmers.se/˜lada

DES — Detached-Eddy Simulations

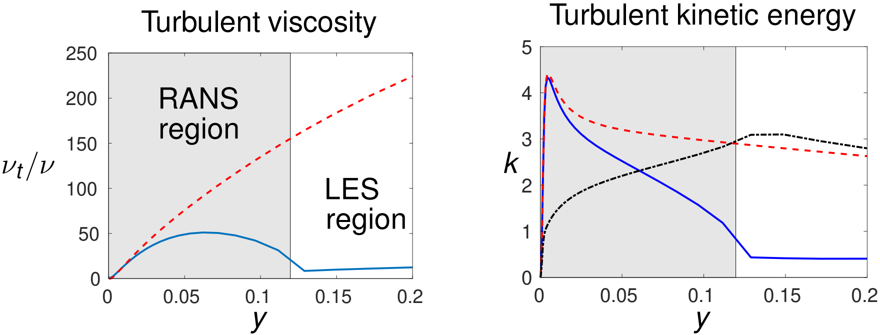
- Problem:
  - the flow in the RANS region is highly unsteady (i.e. URANS)
  - this means that RANS turbulence models (developed for steady flow) are not accurate



—— : DES; – – ; 1D steady RANS; – · – : DES resolved k.

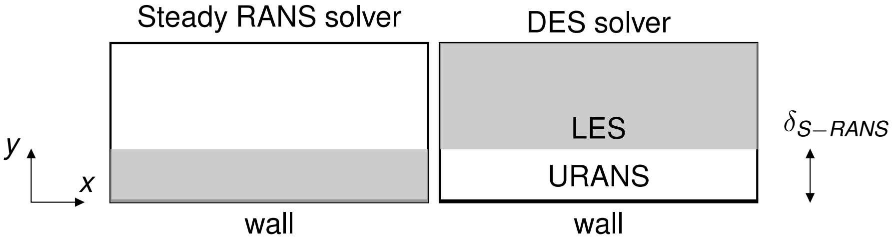
DES — Detached-Eddy Simulations

- Problem:
  - the flow in the RANS region is highly unsteady (i.e. URANS)
  - this means that RANS turbulence models (developed for steady flow) are not accurate
- Solution:
  - solve the steady equations in the RANS region



: DES; ; 1D steady RANS; : DES resolved k.

Two solvers in the entire domain

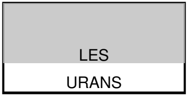


Grey color indicates the solver that drives the flow

Drift terms are added in White regions

Steady RANS solver

DES solver



$$S_i^{RANS} = \frac{\langle v_i^{LES} \rangle_T - \langle v_i^{RANS} \rangle_T}{\Delta t} \qquad S_i^{LES} = \frac{\langle v_i^{RANS} \rangle_T - \langle \bar{v}_i^{LES} \rangle_T}{\Delta t},$$

Subscript T indicates integration over time T

$$\langle \phi(t) \rangle_T = \frac{1}{T} \int_{-\infty}^{t} \phi(\tau) \exp(-(t-\tau)/T) d\tau \Rightarrow$$

$$\langle \phi \rangle_T^t \equiv \langle \phi \rangle_T = a \langle \phi \rangle_T^{t-\Delta t} + (1-a)\phi^t$$

$$a = \exp(-\Delta t / T).$$

WHY IS THE RANS SOLVER STEADY?

Steady RANS solver

$$S_i^{RANS} = \frac{\langle v_i^{RANS} \rangle_T - \langle v_i^{RANS} \rangle_T}{\Delta t}$$

- The RANS solver is called every 10^{th} timestep (can probably be called less often)
- The solution in the RANS solver stays steady when the drift term, S_i^{RANS} is steady (constant in time)
- If the integration time T is too small, there will slightly different steady RANS flow every 10^{th} timestep
    - Solution: make the steady RANS solver unsteady but use the large timestep, i.e. $10\Delta t_{DES}$

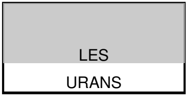
Previous Work

- The present method is similar to those in [1, 2, 3]. The main differences are that
  - In [1, 3] they use one additional drift terms in the LES momentum equations to control resolved Reynolds stresses
  - They include drift terms also in the k and ε equations [1] or the k equation [3].
  - In [1, 3] they include five tuning constants in all drift terms. I have one (T).

Turbulence Models

Steady RANS solver

DES solver



- EARSM (Explicit Algebraic Stress Model) [4] coupled to Wilcox $k-\omega$ model [5]

- DES $k-\omega$ model
- Lengthscale in dissipation term of the k eq.is taken from the IDDES model [6, 7]

RANS and DES turbulence models

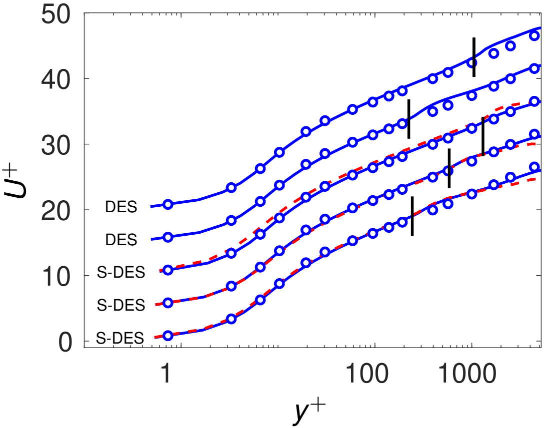
NUMERICAL METHOD: CALC-LES & CALC-BFC

- CALC-LES [8]: DES solver
  - Incompressible finite volume method
  - Pressure-velocity coupling treated with fractional step
  - Central differencing scheme for momentum eqns
  - Hybrid 1^{st} order upwind/2^{nd} order central scheme k & ω eqns.
  - 2^{nd}-order Crank-Nicholson for time discretization
- CALC-BFC [9]: RANS solver, called every 10^{th} timestep
  - Incompressible finite volume method
  - SIMPLEC
  - MUSCL: 2nd order bounded upwind scheme for momentum eqns
  - Hybrid 1^{st} order upwind/2^{nd} order central scheme k & ω eqns.

First Test Case: Channel flow

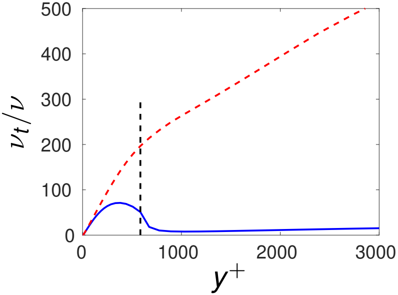
- Reynolds number is $Re_\tau = 5\,200$.
- A $32 \times 96 \times 32$ mesh is used
- $x_{max} = 3.2$, $z_{max} = 1.6$, 15% stretching in y direction

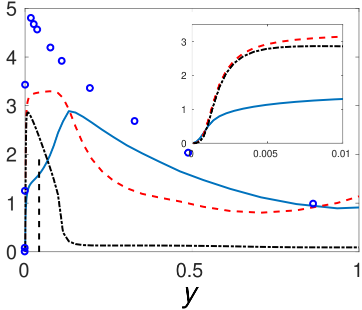
Channel flow: Velocity



$T = 50\delta/U_b$ ——— : DES; - - : RANS; ○: DNS. Vertical black lines show DES interface.

CHANNEL FLOW: TURBULENT QUANTITIES

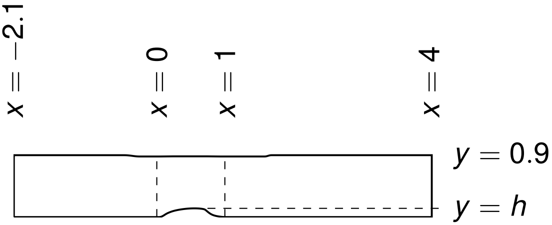




○: DNS [10]; — · —: DES, resolved turbulence

——: DES solver; – –: RANS solver. Vertical black lines show DES interface.

Second Test Case: Hump flow



The domain of the hump. $z_{max} = 0.2$.

- The Reynolds number of the hump flow is $Re_c = 936\,000$.
- The mesh has $386 \times 120 \times 32$ cells (x, y, z)
- Grid from NASA workshop.[1]
- Inlet is located at $x = -2.1$ and the outlet at $x = 4.0$,

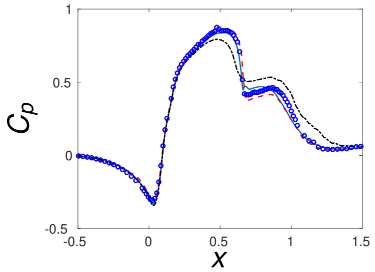
[1] https://turbmodels.larc.nasa.gov/nasahump_val.html

Hump flow: Numerical Issues

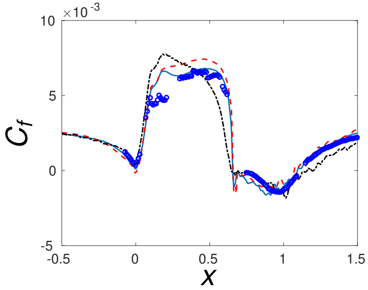
Steady RANS solver

- The drift term in the RANS solver in the LES region (white region) causes unphysical oscillations in the skinfriction
- The problem was traced to the source term in the pressure correction equation, the continuity error $\dot{m}$
- Hence, $\dot{m}$ was set to zero in the LES region.
- As a consequence, the RANS velocity field is driven by the drift term, but the RANS pressure is not correct in this region

Hump flow: C_p & C_f



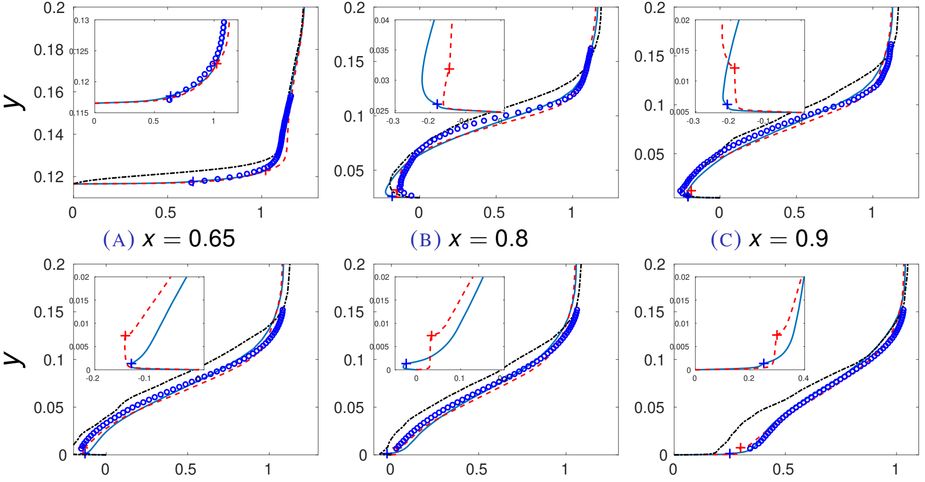
(A) Pressure coefficient.



(B) Skinfriction.

$T = 20h/U_{in}$. ▬▬ : S-DES, $j_0 = 33$; ‐ ‐ : S-DES, $j_0 = 53$; ‐ · ‐ : DES

HUMP FLOW: VELOCITIES



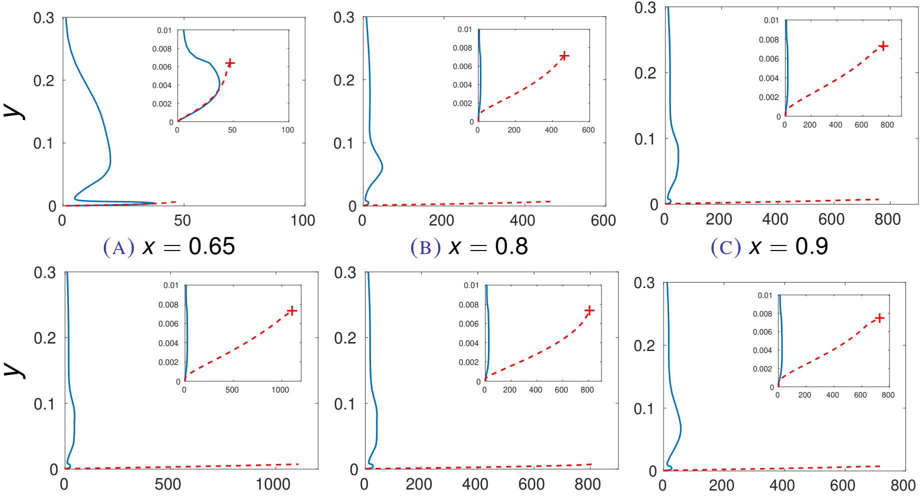
(A) $x = 0.65$ (B) $x = 0.8$ (C) $x = 0.9$

(D) $x = 1.0$ (E) $x = 1.1$ (F) $x = 1.3$

: S-ZDES, $j_0 = 33$; : S-ZDES, $j_0 = 53$; : DES; ○: exp [11, 12];

+,+: DES interface.

TURBULENT VISCOSITY (EARSM). S-ZDES, $j_0 = 53$

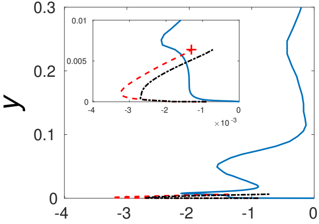


(A) $x = 0.65$ (B) $x = 0.8$ (C) $x = 0.9$

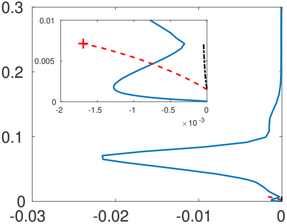
(D) $x = 1.0$ (E) $x = 1.1$ (F) $x = 1.3$

—— : DES solver; - - : RANS solver; +: DES interface.

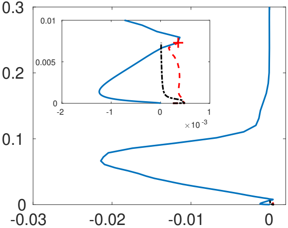
HUMP FLOW: SHEAR STRESSES. S-ZDES, $j_0 = 53$



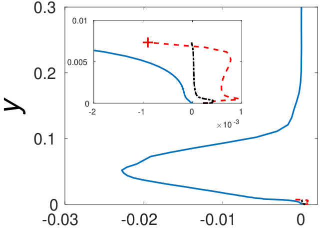
(A) $x = 0.65$ $\times 10^{-3}$



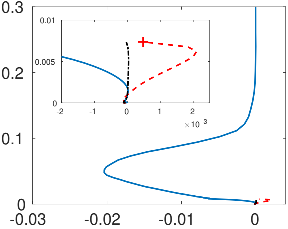
(B) $x = 0.8$



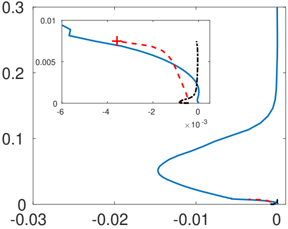
(C) $x = 0.9$



(D) $x = 1.0$



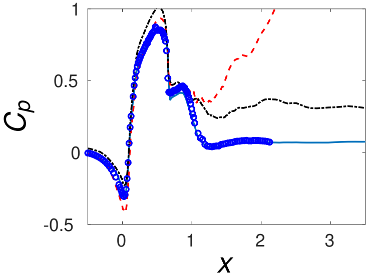
(E) $x = 1.1$



(F) $x = 1.3$

: DES solver, resolved; : RANS solver; : DES solver, modeled.

HUMP FLOW: WRONG p IN RANS REGION



Pressure coefficient. ——— : LES; – – : RANS, $j_0 = 53$; –·–: RANS, $j_0 = 33$

CONCLUSIONS

- A new steady RANS coupled to DES (S-ZDES) is proposed.
- Very good results …
- but the hump results are maybe/probablby contaminated by a numerical fix
- Drawback: it is dependent on the lower limit of integration time, T for the hump flow
  - $T = 10h/U_{in}$ too small (h is hump height)
  - $T = 20$ and 50 give indentical results
  - For $T = 100$ we must more than double developing+sampling time to $345 + 345$ ($7.3 + 7.3$ throughflow times)

References I

[1] H. Xiao and P. Jenna.
A consistent dual-mesh framework for hybrid LES/RANS modeling.
Journal of Computational Physics, 231:1848–1865, 2012.

[2] B. de Laage de Meux, B. Audebert, R. Manceau, and R. Perrin.
Anisotropic linear forcing for synthetic turbulence generation in large eddy simulation and hybrid RANS/LES modeling.
Physics of Fluids A, 27(035115), 2015.

[3] R. Tunstall, D. Laurence, R. Prosser, and A. Skillen.
Towards a generalised dual-mesh hybrid les/rans framework with improved consistency.
Computers & Fluids, 157:73–83, 2017.

References II

[4] S. Wallin and A. V. Johansson.
A new explicit algebraic Reynolds stress model for incompressible and compressible turbulent flows.
Journal of Fluid Mechanics, 403:89–132, 2000.

[5] D. C. Wilcox.
Reassessment of the scale-determining equation.
AIAA Journal, 26(11):1299–1310, 1988.

[6] M. L. Shur, P. R. Spalart, M. Kh. Strelets, and A. K. Travin.
A hybrid RANS-LES approach with delayed-DES and wall-modelled LES capabilities.
International Journal of Heat and Fluid Flow, 29:1638–1649, 2008.

References III

[7] S. Arvidson, L. Davidson, and S.-H. Peng.
Interface methods for grey-area mitigation in turbulence-resolving hybrid RANS-LES.
International Journal of Heat and Fluid Flow, 73:236–257, 2018.

[8] L. Davidson.
CALC-LES: A Fortran code for LES and hybrid LES-RANS.
Technical report, Division of Fluid Dynamics, Dept. of Mechanics and Maritime Sciences, Chalmers University of Technology, Gothenburg, 2018.

[9] L. Davidson and B. Farhanieh.
CALC-BFC: A finite-volume code employing collocated variable arrangement and cartesian velocity components for computation of fluid flow and heat transfer in complex three-dimensional geometries.

References IV

Rept. 95/11, Dept. of Thermo and Fluid Dynamics, Chalmers University of Technology, Gothenburg, 1995.

[10] M. Lee and R. D. Moser.
Direct numerical simulation of turbulent channel flow up to $Re_\tau \approx 5200$.
Journal of Fluid Mechanics, 774:395–415, 2015.

[11] D. Greenblatt, K. B. Paschal, C.-S. Yao, J. Harris, N. W. Schaeffler, and A. E. Washburn.
A separation control CFD validation test case. Part 1: Baseline & steady suction.
AIAA-2004-2220, 2004.

[12] D. Greenblatt, K. B. Paschal, C.-S. Yao, and J. Harris.
A separation control CFD validation test case Part 1: Zero efflux oscillatory blowing.
AIAA-2005-0485, 2005.