Turbulent flow over backward facing step

- Problem description:

A backward facing step of height 9.6mm is subject to freestream air (at STP) flow at velocity 7.72m/s. The transient development of flow structure downstream of the step is simulated using incompressible SA-DDES. OpenFoam v23 and SU2 v7 are used to simulate the flow. The flow domain is chosen based on the experimental work by Jovic and Driver (1994) and subsequent DNS studies by Le et al (1997). Schematic of the flow domain is shown below:

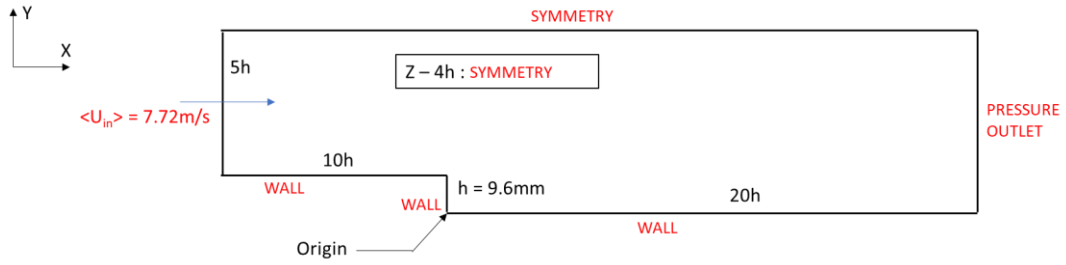

Fig.1 Z-plane of 3D domain and boundary conditions for backward facing step flow simulation

In the Z-plane view shown in Fig. 1, the scripts in red mention the boundary condition of the adjacent side. Note that the width of the domain is 4*h (h = 9.6mm) and is applied symmetry boundary conditions on either side. The Reynolds number based on the step height at edge of the step, Re_h = 5000.

- Mesh description

Mesh for the domain is generated using gmsh. 3 sets of mesh corresponding to varying y^+ values are generated for the simulation. The boundary layer is modeled using SA turbulence model and the first cell y^+ value in the models are constrained within 5. Table 1 gives the list of mesh and time step parameters considered for each y^+ cases. Meshed domains are shown in Figs. 2-4.

Table 1. Mesh and time step parameters

y^+	C_f	u_τ (m/s)	dy (mm)	dt (ms)	Grid Count
5	0.0124	0.61	0.12	0.1	237,120
2.3	0.0124	0.61	0.056	0.04	409,120
1	0.0124	0.61	0.024	0.02	780,120

$C_f = (2\log_{10}(Re_h) - 0.65)^{-2.3}$: Skin friction coefficient using Schlichting model

$\tau_{wall} = \frac{1}{2}\rho_\infty u_\infty^2 C_f$: Wall shear stress

$u_\tau = \sqrt{\frac{\tau_{wall}}{\rho_\infty}} = u_\infty\sqrt{\frac{C_f}{2}}$: Frictional velocity used for turbulent boundary layer scaling

$dy = \frac{y^+ \nu_\infty}{u_\tau}$: First cell height from scaled wall-normal distance

$dt \leq \frac{dy}{u_\tau} * 0.5$: Time step estimated by Nyquist criterion

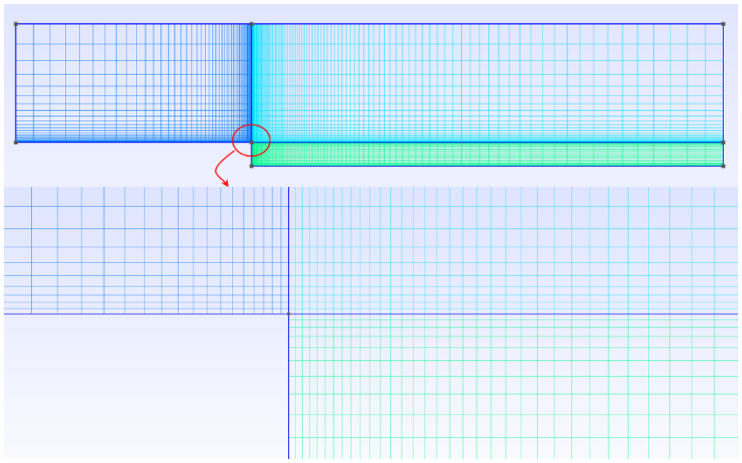

Fig. 2 Mesh domain for $y^+ = 5$

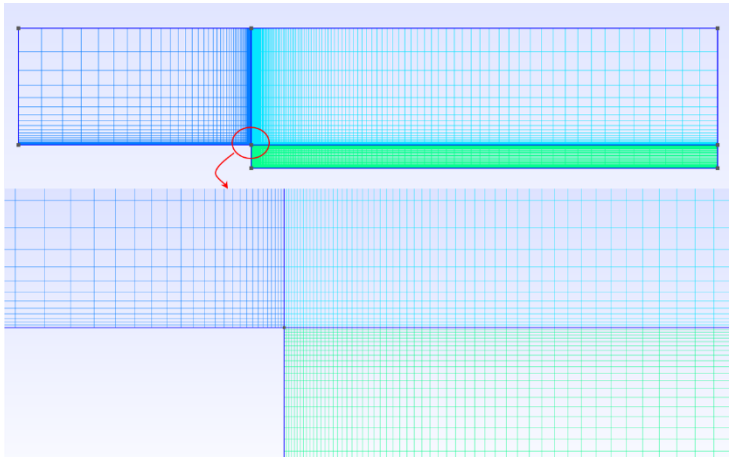

Fig. 3 Mesh domain for $y^+ = 2.3$

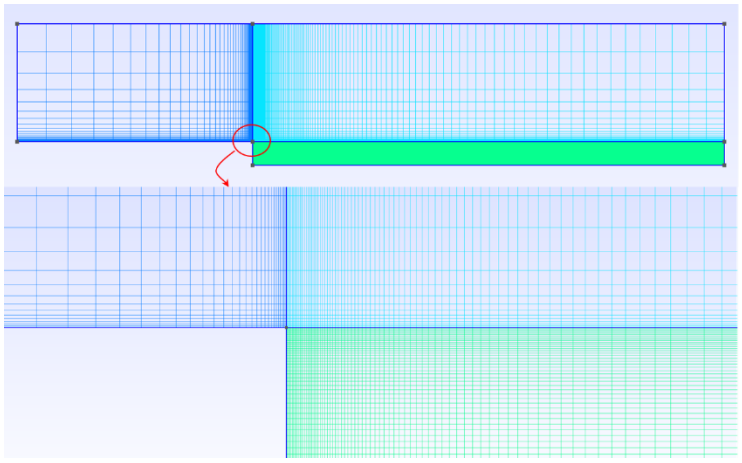

Fig. 4 Mesh domain for $y^+ = 1$

It is to be noted that 40 grid counts are used in Z-direction for all mesh domains.

- Simulation Setup

  Standard SA model is used for near-wall turbulence modeling while Smagorinsky LES sub-grid scale (SGS) model is implemented at wall-normal distances larger than a value specified from default wall-distance coefficient. (Refer to Molina et al (2017) for SU2 formulations).

  In SU2, 2nd order dual-time stepping scheme is used for time marching method with 5 inner iterations per time step. Total simulation duration is 400ms which corresponds to roughly 10 flow passes across stream wise direction. Results are saved every 2ms.

- Sample Results

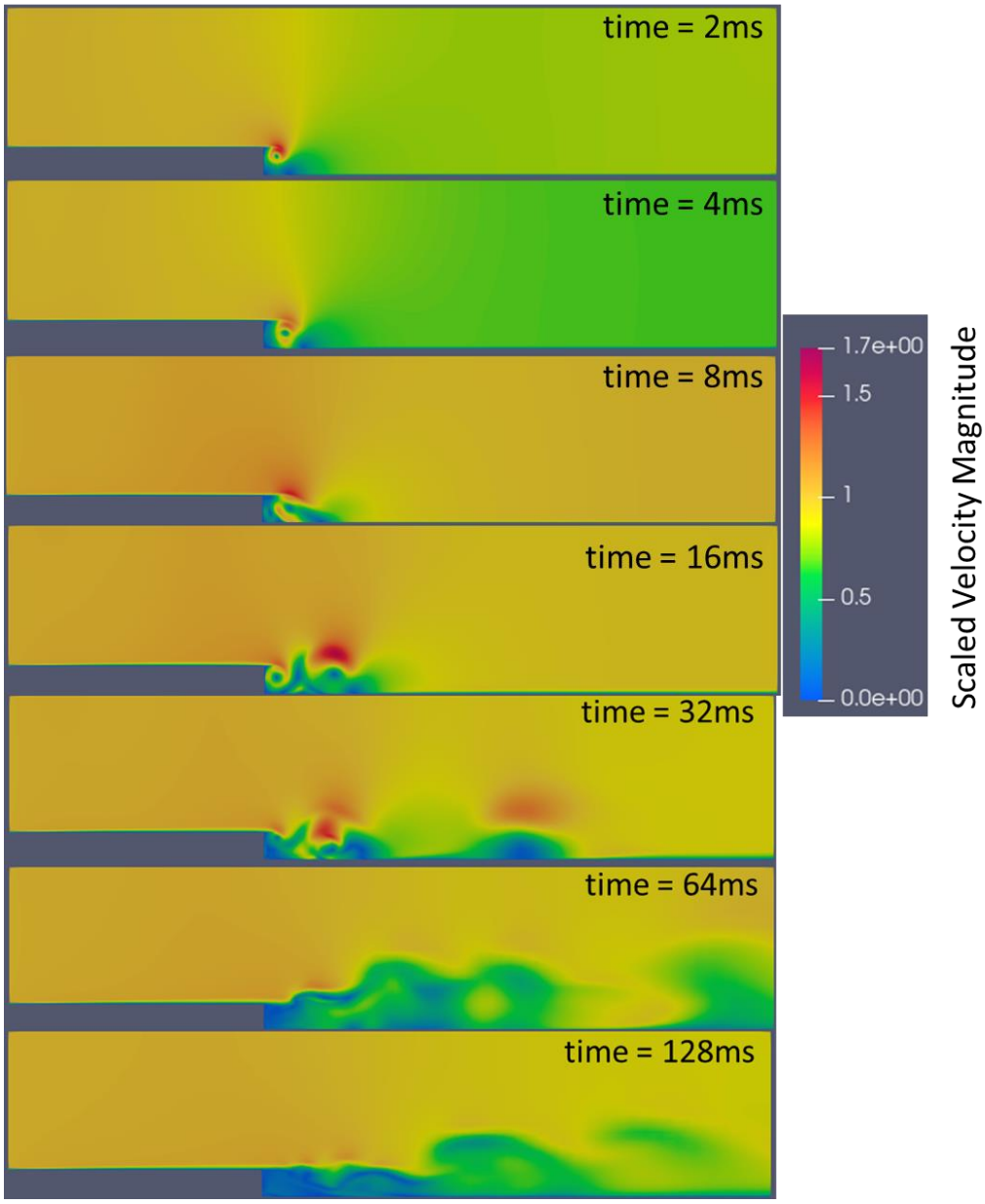

Fig. 5 Results for mesh domain with $y^+ = 5$

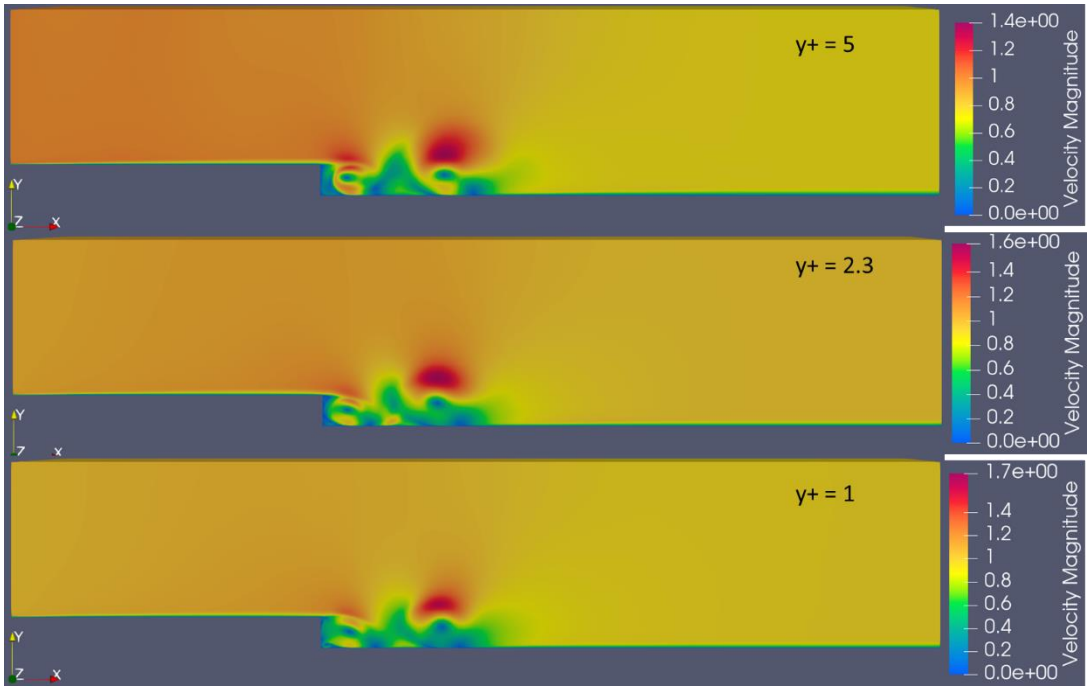

Fig. 6 Results for mesh domain at t = 20ms (contours scaled with freestream velocity)

- References

Jovic, S., & Driver, D. M. (1994). *Backward-facing step measurements at low Reynolds number, Re (sub h)= 5000* (No. NASA-TM-108807).

Le, H., Moin, P., & Kim, J. (1997). *Direct numerical simulation of turbulent flow over a backward-facing step.* Journal of fluid mechanics, 330, 349-374.

Molina, E., Spode, C., Annes da Silva, R. G., Manosalvas-Kjono, D. E., Nimmagadda, S., Economon, T. D. & Righi, M. (2017). *Hybrid RANS/LES calculations in SU2.* In 23rd AIAA Computational fluid dynamics conference (p. 4284).