

# Turbulence Modelling

## Computational Modelling of Turbulent Air Flow in the High-Speed Leg of the Virginia Tech Stability Wind Tunnel

*Module Leader: Tamás István Józsa*

### Introduction

Experimental and computational fluid dynamics (EFD and CFD) are the workhorses of product design, research, and development (R&D) in multiple sectors, including the aerospace and automotive industries. CFD based on the Reynolds-averaged Navier-Stokes (RANS) equations are particularly popular because they strike a balance between the reliability of the predictions and computational cost. However, EFD and RANS-based CFD measurements of quantities of interest (such as lift and drag coefficients) are often burdened by more than 10% relative difference. Such high uncertainty manifests as a strong limitation in R&D activities because improvements below the uncertainty level cannot be quantified. It is anticipated that mismatching flow conditions and turbulence modelling errors have a strong impact on the difference between EFD and CFD, but their contributions remain unclear.

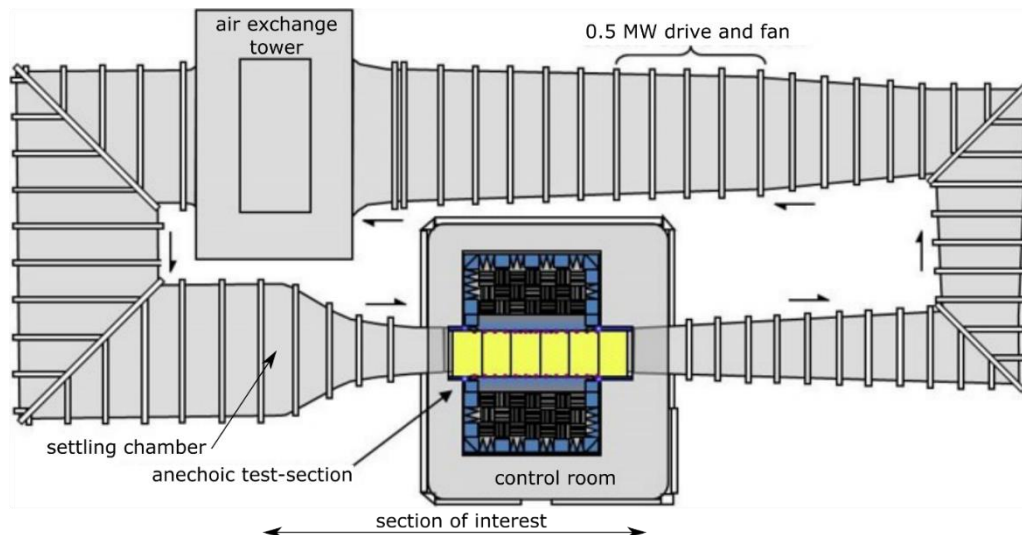


Figure 1 Schematic of the wind tunnel circuit and the section of interest including the test section. In this closed-circuit tunnel, air flow is driven by a fan placed in the low-speed leg of the tunnel. From the fan, air flows towards the settling chamber through two turning vanes. In the settling chamber, air passes through a porous screen to weaken secondary currents and the associated asymmetric flow field caused by the turning vanes. Thereafter, air rushes through a converging section to arrive to the test section where laminar-turbulent transition is triggered by a step. The section of interest includes part of the settling chamber and the entirety of the test section with an extruded region before the outlet replacing the diffuser.

In this assignment, **incompressible and isothermal** RANS<sup>1</sup> simulations of the high-speed leg of the Virginia Tech Stability Wind Tunnel will be carried out using the commercial ANSYS Fluent software. The wind tunnel is described and preliminary simulation results are shown in the study of (M. Szőke, 2020). The wind tunnel circuit and the section of interest is depicted in Figure 1. The purpose of the investigations is to quantify the impact of different turbulence models and inlet boundary conditions on the flow field in the test section. In order to gain meaningful insights, experimentally realisable flow conditions must be used for the computations.

---

<sup>1</sup> RANS usually refers to steady state simulations whereas unsteady models are often described as URANS. In this assignment, only steady state RANS simulations are considered.

## Tasks

During this assignment, a personalised simulation scenario is allocated to everyone. The personalised flow conditions are summarised in Table 1 accompanied by Table 2 where further details are provided. Meshes can be downloaded from [here](#). **WARNING: meshes were created in inches!**

1. Calculate the  $u^{\text{ref}}$  velocity, reference density ( $\rho^{\text{ref}}$ ), and kinematic viscosity ( $\nu$ ) at the location of the Pitot-static probe and justify the assumption of incompressible and isothermal flow.
2. Calibrate your simulation and determine the simulation parameters needed to match the experimental setup using the very coarse mesh. The whole cohort should agree on the numerical settings which must be kept the same in every simulation. Pay attention to the nondimensional wall distance and discuss which wall function (when needed) would be most appropriate.
3. Carry out the computations and check that flow properties align with the experimental case. Ensure the quality of the simulations by monitoring mass imbalance and reference quantities convergence in addition to residuals. Perform a grid convergence study using at least three grids including the very coarse mesh and assess the spatial resolution of the simulation.
4. Visualise the flow field focusing on the pressure and the velocity field and describe the key flow features including vortical motions. Comment on how the results compare with your expectations based on the literature. The direct numerical study of (S. Pirozzoli, 2018) might serve as a good starting point.
5. Demonstrate that a spatially developing boundary layer is present in the test section with self-similar profiles with increasing streamwise coordinate. Compare the results to the law of the wall (including both the viscous sublayer and the log-law) and to [direct numerical simulation data](#) of the flat plate (P. Schlatter, 2010).
6. Investigate how the velocity profile compares with reference data in the symmetry plane of the test section and near the corners and explain your observations based on the computed structure of the Reynolds stress tensor.
7. The variables of interest are listed [here](#). Export and format the data for all grid levels and upload it to [this shared folder](#). The uploaded files must be formatted to match exactly the format of the provided templates and file headers must be modified to show student and simulation details.
8. Consider the test section and carry out calculations to determine the spatial resolution and time step size for a DNS or LES study of your case (as indicated in Table 1). Estimate the memory requirements and computational time needed to perform these simulations. Comment on how simulation parameters should be adjusted if an aerofoil was placed in the test section with relatively high angle of attack. (Running scale-resolved simulations is not required.)
9. Utilise the data of another case where only one parameter is different from your settings and carry out a detailed comparison. Beyond visualisation, quantify the differences, for example, by highlighting the size of the volume where the mean velocities differ by more than 5%, by reporting the locations and values of minimum and maximum differences, etc.

## Hints

- Tasks 1: The necessary calculations rely on the ideal gas law and the definition of the stagnation pressure, stagnation temperature and Mach numbers. Thereafter, the inlet flow condition can be estimated based on the simplified continuity equation establishing the product of the cross-sectional area ( $A$ ) and the bulk velocity ( $u_b$ ) is constant ( $A \cdot u_b = \text{constant}$ ). The simplified nonlinear governing equations can be solved iteratively (e.g., based on a Matlab script).
- Task 2: Multiple simulations should be carried out with inlet velocities above and below the estimated  $u_b$  and zero pressure outlet. Based on the simulation results, a curve can be fitted to establish the relationship between the inlet flow condition and the local reference velocity  $u^{\text{ref}}$ .

Finally, the necessary inlet condition should be found based on interpolation to match the reference velocity. The outlet pressure should be offset to match  $p^{\text{ref}}$ .

## Comments

- Multiple tasks require scripts which need to be developed once and can be re-used thereafter. You may work together and distribute parts of the allocated tasks smartly to save time.
- Consider creating a private github repository where the members of the cohort can share those preprocessing, processing, and postprocessing scripts which are useful for everyone.
- Feel free to utilise artificial intelligence tools, such as ChatGPT and Grammarly to accelerate script development and enhance your writing. However, do not forget to mention the applied tools and specifying how they were used in the “Acknowledgment” section of your report.

## Report requirements

Your report should contain about 3000 words ( $\pm 10\%$ ), excluding abstract, table of content, list of figures, etc. and any appendices). You should not include more than 10 figures and tables in total so consider using stacked figure format (i.e., subfigures). Ensure that figures are readable in printed format (i.e., without excessive zooming, etc.) The report should include the following parts:

1. Introduction, providing (i) motivation for simulating wind tunnels, (ii) key findings from the literature related to the topic to date, and (iii) the aim and objectives of the present report. The literature review should describe the key flow features in low speed closed circuit wind tunnels.
2. Methods, describing the details of the employed techniques, e.g., Fluent settings, boundary conditions, etc. The reported work should be reproducible.
3. Results and discussion, analysing the results, computations, and visualisations. Draw on the literature to make comparisons, comment on trends and whether you can match these. Practice critical evaluation of your findings based on literature data and comment both on similarities and differences. The results ideally
4. Conclusion, summarising your findings and the learning outcome. Consider referring to the introduction (motivation and/or aim) and comment on how your work meets the specified aim.

## References

- M. Szőke, V. V.-P. (2020). Developing a numerical model of the virginia tech stability wind tunnel for uncertainty quantification based on real-world geometry. *AIAA Scitech Forum*. Orlando, Florida, USA.
- P. Schlatter, R. Ö. (2010). Assessment of direct numerical simulation data of turbulent boundary layers. *Journal of Fluid Mechanics*, 659, 116-126.
- S. Pirozzoli, D. M. (2018). Turbulence and secondary motions in square duct flow. *Journal of Fluid Mechanic*, 840, 631-655.

Table 1 Key settings of individualised simulations. **Computations must be carried out with constant viscosity and density** inferred based on Table 2. CFC – corner flow correction; QPS – quadratic pressure-strain; FD – inlet data generated based on a separate duct flow simulation; CP – custom profile generated based on the script available from this GitHub repository: [https://github.com/jozsait/VT\\_CRWT\\_inlet\\_generator](https://github.com/jozsait/VT_CRWT_inlet_generator) (the generated profile might need coordinate transformation); unif. – uniform; FD – fully developed; TI – turbulence intensity.

Student ID	Ref. Mach number	Turbulence model	Inlet mean flow	Inlet turbulence	Calculation case
324733/1	0.082	SA	unif.	TI = 0 %	LES
407425/1	0.136	SA	unif.	TI = 0 %	DNS
415248/1	0.193	SA	unif.	TI = 0 %	LES
431378/1	0.082	SA CFC	unif.	TI = 0 %	DNS
419764/1	0.136	SA CFC	unif.	TI = 0 %	LES
421612/1	0.193	SA CFC	unif.	TI = 0 %	DNS
431223/1	0.082	k- $\epsilon$ standard	unif.	TI = 0 %	LES
432303/1	0.136	k- $\epsilon$ standard	unif.	TI = 0 %	DNS
433730/1	0.193	k- $\epsilon$ standard	unif.	TI = 0 %	LES
395162/1	0.082	k- $\epsilon$ RNG	unif.	TI = 0 %	DNS
424491/1	0.136	k- $\epsilon$ RNG	unif.	TI = 0 %	LES
418861/1	0.193	k- $\epsilon$ RNG	unif.	TI = 0 %	DNS
385747/1	0.082	k- $\epsilon$ realizable	unif.	TI = 0 %	LES
415206/1	0.136	k- $\epsilon$ realizable	unif.	TI = 0 %	DNS
407439/2	0.193	k- $\epsilon$ realizable	unif.	TI = 0 %	LES
417950/1	0.082	k- $\omega$ standard	unif.	TI = 0 %	DNS
367896/1	0.136	k- $\omega$ standard	unif.	TI = 0 %	LES
406274/1	0.193	k- $\omega$ standard	unif.	TI = 0 %	DNS
402993/1	0.082	k- $\omega$ SST	unif.	TI = 0 %	LES
438545/1	0.136	k- $\omega$ SST	unif.	TI = 0 %	DNS
389161/1	0.193	k- $\omega$ SST	unif.	TI = 0 %	LES
404566/2	0.082	k- $\omega$ SST CFC	unif.	TI = 0 %	DNS
404591/1	0.136	k- $\omega$ SST CFC	unif.	TI = 0 %	LES
398729/1	0.193	k- $\omega$ SST CFC	unif.	TI = 0 %	DNS
432070/1	0.082	k- $\omega$ GEKO CFC	unif.	TI = 0 %	LES
405209/1	0.136	k- $\omega$ GEKO CFC	unif.	TI = 0 %	DNS
425161/1	0.193	k- $\omega$ GEKO CFC	unif.	TI = 0 %	LES
431116/1	0.082	RSM	unif.	TI = 0 %	DNS
418173/1	0.136	RSM	unif.	TI = 0 %	LES
436688/1	0.193	RSM	unif.	TI = 0 %	DNS
419447/1	0.082	RSM QPS	unif.	TI = 0 %	LES
074999/2	0.082	k- $\omega$ SST CFC	unif.	TI = 1 %	DNS
447400/1	0.082	k- $\omega$ SST CFC	FD	FD	LES
444799/1	0.082	k- $\omega$ SST CFC	FD	TI = 0 %	DNS
448014/1	0.082	k- $\omega$ SST CFC	CP	TI = 0 %	LES
t1868227	0.082	turbulence model and software of your choice	unif.	TI = 0 %	LES
s447400	0.082				DNS
s448014	0.082				LES

Table 2 Summary of experimental pressure and temperature measurements needed for the calculation of the personalised flow conditions. Stagnation and static pressures were determined using a Pitot-static probe placed at  $x=3.502$  m,  $y=-0.608$  m,  $z=0$  m as shown in Figure 2. Reference quantities measured at this location are distinguished by the “ref” superscript. Notation:  $u$  – mean streamwise velocity [m/s];  $a$  – speed of sound [m/s];  $\nu$  – kinematic viscosity [m<sup>2</sup>/s];  $Ma$  – Mach number;  $T_{\text{tot}}$  – stagnation temperature.

Reference velocity-viscosity ratio	Reference Mach number	Local stagnation temperature	Local stagnation pressure	Local static pressure
$u^{\text{ref}}/\nu$	$Ma^{\text{ref}} = u^{\text{ref}}/a^{\text{ref}}$	$T_{\text{tot}}^{\text{ref}}$	$p_{\text{tot}}^{\text{ref}}$	$p^{\text{ref}}$
[1/m]	[-]	[K]	[Pa]	[Pa]
$1.7 \times 10^6$	0.082	297.2	94450	94009
$2.8 \times 10^6$	0.136	298.4	94359	93147
$3.9 \times 10^6$	0.193	300.4	94242	91832

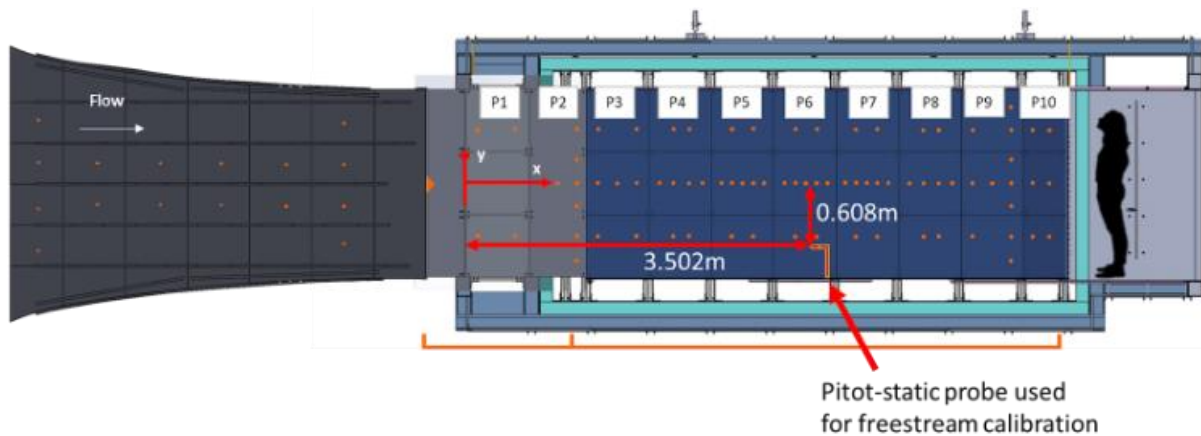


Figure 2 Schematic showing the location of the Pitot-static probe measurement used for freestream flow calibration in the wind tunnel experiments.