

# 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.

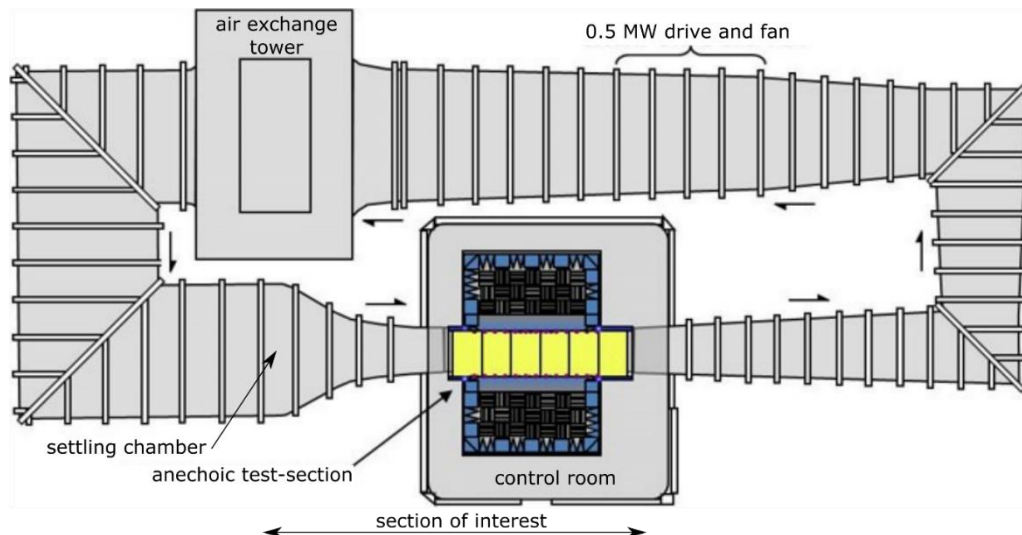


Figure 1 Schematic of the wind tunnel circuit and the section of interest including the test section. In this closed-circuit tunnel, air flow is driven by a fan placed in the low-speed leg of the tunnel. From the fan, air flows towards the settling chamber through two turning vanes. In the settling chamber, 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, geometries, 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, you are going to work in groups as summarised in Table 1 accompanied by Table 2 where further flow condition details are provided.

Download link for [as-designed meshes](#) and [as-designed inflow generator](#).

Download link for [as-built meshes](#) and [as-built inflow generator](#).

Level 10 is the coarsest and level 1 is the finest mesh.

**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. Every group will need to agree on the numerical settings which must be kept unchanged to facilitate comparisons. Pay attention to the nondimensional wall distance ( $y^+$ ) and discuss which wall treatment is 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 level7, level6, and level5 meshes and assess the spatial resolution.
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 simulation study of (S. Pirozzoli, 2018) might serve as a good starting point.
5. Investigate how the velocity profile compares with reference data from the literature 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.
6. Consider the **baseline scenario with default k-omega SST turbulence model, uniform inflow condition, as-designed geometry, level 7 mesh, and reference Mach number 0.082**. The groups will need to generate data for multiple cases where at least one parameter is different compared to the baseline set and carry out detailed comparisons to quantify the impact of various input parameters. You may work together to assemble a simulation matrix for the whole cohort and thus avoid running the same simulations twice. Beyond visualisation, quantify the differences, for example, by highlighting the size of the volume where the mean velocities differ by more than 5%, or by reporting where the flow field is most uncertain due to turbulence modelling, etc. **Group members shall run cases to study comparatively the impact of RANS turbulence models, reference Mach number, uniform inflow vs. non-uniform inflow, as-designed vs. as built geometry, inflow turbulence intensity, etc. Extended versions of the turbulence models can be also considered, such as corner flow correction.**

## Optional tasks

1. 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).

## 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). The inlet boundary condition should be set with uniform mean velocity and 0% turbulence intensity.
- 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.
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 Mechanics*, 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. (PT) → part-time student

Group	Initials
1	ThoB
	VB
	SB (PT)
	ThaB
	PB
	JB
2	QBM
	HC
	YNRKC
	SYDC
	SC
	BC
3	JD
	GG
	KSKB
	PLC
	JMC
	NKM
4	AN
	KO
	SPG
	JYP
	MP
	NR
5	SVS
	SV
	KV
	YW
	SW (PT)
	SY
	TZ

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
-------------------	-------	-------	-------	-------

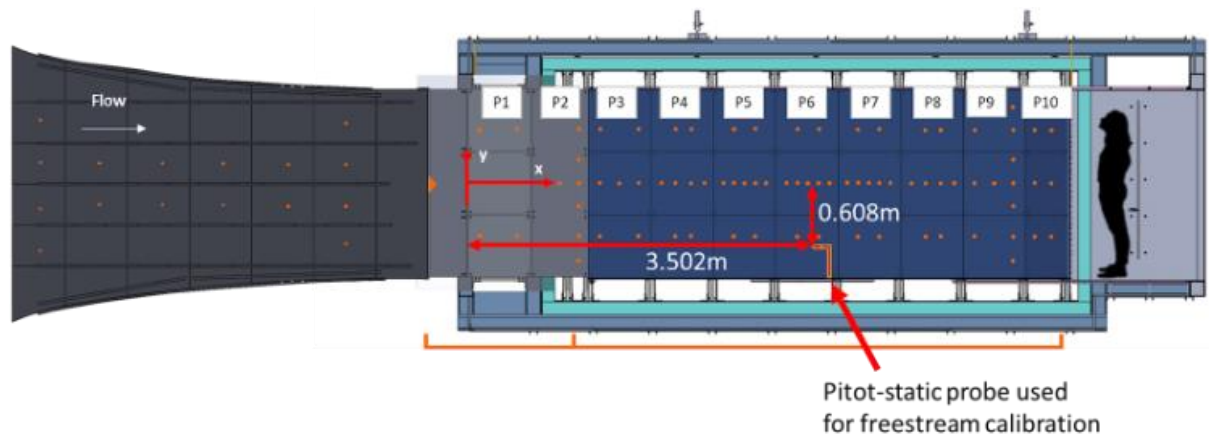


Figure 2 Schematic showing the location of the Pitot-static probe measurement used for freestream flow calibration in the wind tunnel experiments.