CFD modeling of the Poul la Cour Tunnel

Niels N. Sørensen¹, A. S. Olsen¹, C. Bak¹, M. Gaunaa¹, J.S. Beckerlee¹, A. Fischer¹, R.F. Mikkelsen¹, S. Ildvedsen¹

¹Technical University of Denmark, Frederiksborgvej 399, 4000 Roskilde
E-mail: nsqr@dtu.dk

Abstract. We analyze the new Danish Poul la Cour Tunnel (PLCT) facility using 2D and 3D Computation Fluid Dynamics simulations, with the aim to address the need for wind tunnel corrections. The assessment of tunnel corrections is based on 2D free simulations representing undisturbed airfoil flow, 2D tunnel configurations and a full 3D tunnel setup using laminar/turbulent Improved Delayed Detached Eddy Simulations. By comparing lift, drag and pressure distributions and detailed flow patterns, the influence of the 2D and 3D tunnel is evaluated. In the present study a NACA 63-418 airfoil is investigated in the tunnel.

1. Introduction
When measuring airfoil performance in a wind tunnel, corrections must be made to account for the difference between the tunnel conditions and the idealized free conditions where the airfoil operates in an infinite uniform velocity field. Classical tunnel corrections dates back to Glauert [1] and typically accounts for solid blockage, streamline curvature, wake blockage and so called buoyancy effects due to pressure gradients in the tunnel [2, 3]. Most of the traditional work on wind tunnel corrections are based on 2D potential flow theory, but the classical corrections have over the last ten to fifteen years been supplemented with Computational Fluid Dynamic (CFD) studies, where viscous effects can be included in the considerations, see e.g. [4, 5, 6].

2. Methodology
We investigate the influence of the wind tunnel geometry on the aerodynamic properties of a wing section by comparing the measured values in the tunnel to a 3D CFD model consisting of the contraction, test section and diffusor part of the PLCT. The 3D model is supplemented by a 2D model of the test section and a 2D free model, to help interpretate the results.

The 3D CFD simulations are performed with the EllipSys3D solver, using time true Improved Delayed Detached Eddy Simulation (IDDES) [7] model based on the $k-\omega$ SST model [8], along with $\gamma - Re_\Theta$ correlation based transition model [9, 10]. The EllipSys3D code is second order accurate in time, using a implicit time stepping with sub-iterations. In the present simulations a time-step of $1.0 \times 10^{-4}$ seconds is applied. The convective terms are approximated by the QUICK scheme by Leonard [11] in the Reynolds Averaged Navier-Stokes (RANS) regions and a 4th order central scheme in the Large Eddy Simulation (LES) regions as suggested by Travin et al [12]. The diffusive terms are discretized using second order accurate central differences. The 2D tunnel and free simulations are both based on steady state RANS $k-\omega$ SST simulations using EllipSys2D.
In the present work we mainly focus on angles of attack where the flow is attached at the central part of the wing. In the attached region the IDDES model reduces to the RANS solution, while the IDDES model is only active downstream of the junction area between the wing and the tunnel walls.

2.1. **Computational grid and boundary conditions**

The grid for the 2D tunnel model is a slice from the 3D tunnel model, while the 2D free model is using an O-mesh with the same surface grid distributions as used in the tunnel simulations. The chordwise resolution of all the meshes is 320 cells around the airfoil, and the wall normal grid spacing is set to $y^+ \sim 1$. For the 2D free simulation the far field boundary is placed 50 airfoil chords away, and 128 cells are used in the normal direction. This is a typical setup for a 2D free simulation, with 40960 cells. For the 2D tunnel simulations the actual geometry from the PLCT tunnel is used, see Fig. 1, embedding the inner O-mesh of 320x128 cells in a outer H-mesh configuration. The test section of the PLCT covered by the H-mesh is 9 meters long with the airfoil pitch center, here $0.4 \times$ chord, 4 meters downstream of the test section inlet. It has 64 cells upstream and 128 cells downstream of the airfoil in the flow direction, and has 192 cells across the channel with a cell height $y^+ < 1$ at the tunnel walls. The total number of cells are 90112 cells. For the 3D simulations, the test section is extended in the upstream direction with the actual contraction geometry to model the acceleration of the flow. Additionally, a downstream exhaust region is added behind the test section to dampen numerical instability at the outlet due to the separated flow generated at the airfoil junction with the tunnel wall. In span-wise direction the tunnel is 2 meters high and the grid has 128 cells stretched towards the wall to obtain a $y^+ \sim 1$ at the top and bottom tunnel walls. The total number of cells for the 3D configuration is 13.9 million cells, which is substantially higher than the 2D simulations.

At all solid walls no-slip conditions are applied, this goes for the wing/airfoil geometry and for the wall surfaces of the wind tunnel. The inlet region is specified by Dirichlet conditions for velocity and turbulence quantities, with a constant values of inlet velocity, $k$ and $\omega$. The outlet is specified as a fully developed region using a Neumann condition. Initially different approaches has been investigated for the 2D tunnel setup. Option 1: Prescribed constant inlet velocity combined with fully developed outlet condition. Option 2: Prescribed constant inlet velocity along with prescribed static pressure in the outlet. Option 3: Prescribed total pressure and velocity direction at the inlet, combined with prescribed static pressure at the outlet. Minimal difference was observed in the airfoil results between the three boundary condition options, and the well proven prescribed constant inlet velocity along with a fully developed flow in the outlet are therefore preferred for all simulations in the present work. For the 2D free simulations, inlet conditions are specified for the main part of the outer O-mesh boundary, while a 90 degrees region downstream of the airfoil centered around the inflow direction is specified as outlet.
3. Results

Based on the described techniques a NACA 633−418 airfoil with a chord length of 1 meter, is studied at a Reynolds number of 3 million. The airfoil loads are evaluated based on viscous stresses and pressure on the wing section, on the tunnel walls and the deficits in the wake of the wing section. The detailed 3D fields available from the simulations directly provides insight to the effects of viscosity, turbulence, and laminar/turbulent transition.

For the 2D free and the 2D tunnel simulations, the desired velocity is directly specified at the inlet. In case of the 3D tunnel configuration the settling chamber velocity is specified and the actual test section velocity is a function of the change in tunnel cross section for the empty section. With the airfoil inserted, we confirm that the pressure rise in the contraction corresponds to the empty tunnel setup, similarly to the approach used in the physical tunnel. The relation between the settling chamber velocity and the predicted velocity in the test section, fits very well with the theoretical contraction ratio of 9 and can to a good approximation be approximated by the following expression:

$$U_{ts} = 9.00 \times U_{sc}$$  \hspace{1cm} (1)

where $U_{ts}$ is the test section velocity and $U_{sc}$ is the settling chamber velocity upstream of the contraction. To have a Reynolds number of 3 million with an airfoil chord of 1 meter assuming a density of 1.205 [kg m$^{-3}$] and a molecular viscosity of $1.82 \times 10^{-5}$ [kg m$^{-1}$ s$^{-1}$] we need a test section chamber velocity of 45.31 [m s$^{-1}$]

3.1. Flow features

Looking at the qualitative features of the computed flow fields, it can be observed that horseshoe vortices are forming near the leading edge of the airfoil at the junction between the wing section and the tunnel floor and ceiling, see Fig. 2 or more clearly in Fig. 3. At small angles of attack at [-4:4] degrees with fully attached flow on the airfoil, a system of counter rotating vortices are formed around the junction, as seen for the 4 degrees angle of attack case in the left side of Fig. 3. At higher angles of attack, flow separation starts to develop near the trailing edge on the suction side of the airfoil at the junction with the solid tunnel walls, see right hand side of Fig.3. In this situation, the horseshoe vortex formed at the leading edge is blended with the trailing edge separation pattern forming a very complex flow field.

The development of the corner separation can be studied in Fig.4, showing the surface streamlines based on the mean flow at four different angles of attack [0, 4, 6, 8] degrees. For the zero degree angle of attack, the flow is fully attached and only a small deviation of the streamline direction from chordwise can be observed near the trailing edge close to the tunnel.
wall, see the top left picture in Fig. 4. Increasing the angle of attack to 4 degrees, a small separated area is developing at each of the two solid tunnel walls near the airfoil trailing edge, see the top right picture in Fig. 4. Finally, at the 6 and 8 degrees angle of attack large corner separations can be observed in both cases, see the bottom left and right pictures in Fig. 4. The reason that the separation is starting to form in the junction region between the tunnel solid walls and the airfoil, is the fact that the momentum of the oncoming fluid needs to overcome the combined action of the adverse pressure gradient on the aft part of the airfoil and the viscous effects from both the airfoil and the tunnel wall acting in this region of the flow. In contrast to the behavior on the suction side of the airfoil, no tendency to formation of flow separation in the corners are observed on the pressure side of the airfoil for positive angles of attack, not shown here. Even though the flow on the pressure side is also exposed to the combined effect of the viscous effects from both the airfoil and the solid tunnel wall in the corner flow, the existence of a favorable pressure gradient on the airfoil in this region prevents the flow from separating. Flow visualizations with tufts in the physical wind tunnel confirms the overall behavior from the computations with regards to the qualitative development of the corner separations, even though further quantitative data would be need to fully validate the computations.

3.2. Lift as function of angle of attack

Two observations can be made when comparing the lift integrated from the airfoil pressure distributions at the central section, between the wind tunnel measurements and the computations, see Fig. 5. The well known over prediction of the max $C_l$ observed in most CFD simulations is present, with the CFD simulations predicting a max $C_l$ of $\sim 1.7$ in contrast to the measured value of $\sim 1.35$. Focusing instead on the attached region from $\sim [\:-5:8\]$ degrees, we observe a good agreement between the two sets of uncorrected data with a difference between the two results of around 2 percent. Comparing the measured uncorrected lift based on the integrated airfoil pressure at the central airfoil section with the prediction from the 2D tunnel configuration, we observe a consistently higher value in the 2D tunnel predictions, see Fig. 6. Comparing the lift based on airfoil pressure between the 2D tunnel configuration and the 2D free configuration, we observe that the 2D tunnel configuration results in a higher lift as expected from analytical tunnel corrections, see Allen and Vincenti et al. [2]. The fact that the difference between the 2D tunnel predictions and the free 2D predictions qualitatively agrees with standard tunnel corrections, but the 2D tunnel predictions result in higher values than the 3D tunnel predictions at high angles of attack $[5:8]$ degrees, indicate that additional effects are present in the 3D setup, see Fig 7. The 3D effects caused by the existence of the horseshoe vortex at the junction between the wing and the tunnel
Figure 4. Surface streamlines at the suction side of the airfoil based of the of the mean flow, for 0, 4, 6, 8 degrees from top left to bottom right. The decreasing laminar flow region at the leading edge of the airfoil as function of increasing is shown in green while the fully turbulent area is shown in red.

Figure 5. Comparison of the uncorrected lift based on airfoil pressure, between measurements and 3D tunnel computations.
Figure 6. Comparison of the uncorrected lift based on airfoil pressure, between measurements, 2D tunnel computations and 2D free computations.

wall along with the fact that we observe an early stall at the trailing edge of the airfoil at the suction side in proximity to the wall, could explain this. Similar observations, both with respect to the lower lift coefficient in 3D and the horseshoe and corner separation were made in the CFD study of Klein et al. [5]. To support this theory the lift distribution is extracted along the span of the airfoil from the 3D tunnel simulations and shown together with the 2D lift value from the tunnel simulation, see Fig. 8. The figure indicates that the overall lift level along the total airfoil span is lower than the one predicted in 2D and that the 3D lift is especially low near the walls. This effect is highly dependent on the state of the boundary layer properties at the time it interacts with the airfoil junction. As the actual boundary layer properties in the PLCT is not yet documented, the agreement between the computed and actual boundary layer cannot be evaluated at present time.

3.3. Drag
In the experiment the drag of the airfoil is determined using a wake rake, as suggested by Jones [13]. In the CFD simulations (2D and 3D) the drag of the airfoil is determined by evaluating the formula of Jones in each point of the grid, and performing a numerical integration simulating a wake rake. The expression for the drag given by Jones is given below:

$$C_d = \frac{1}{G_0} \int \int 2\sqrt{G-P} \left(\sqrt{G_0} - \sqrt{G}\right) dA,$$

where $G_0$ is the total pressure of the upstream undisturbed flow and $G$ is the local value of the total pressure and $P$ is the local static pressure. Besides the wake integration, the pressure part of the drag can be evaluated from the airfoil pressure both in the measurements and the simulations. Looking at Fig. 9 we see that there is generally very good agreement between the measured and computed drag, both for the pressure part of the drag and for the total drag. Comparing the total drag computed for the 2D free setup, to the uncorrected drag value from
Figure 7. Comparison of the uncorrected lift based on airfoil pressure, between measurements, 2D tunnel computations and 3D tunnel computations.

Figure 8. Spanwise variation of the lift along the airfoil section integrated from the airfoil surface pressure from the 3D tunnel simulation, compared to the value from the 2D tunnel simulation for an angle of attack of 4 degrees.
the measurement we observe very good agreement within a few percent for angles of attack between [-3:5] degrees.

3.4. Pressure distributions
The pressure distribution at 4 and 8 degrees angle of attack are compared between measurements and 3D computations in Fig. 10. The first observation is that the predicted distributions are more smooth than the measured ones, which might be due to production imperfections in the airfoil shape and the installation of the pressure tubings. Additionally, both the 4 and the 8 degrees measured pressure distributions show a higher value of $-C_p$ in the trailing edge region indicating separated flow. The agreement is far from perfect, and some error bars will be needed on the measurements to fully establish the degree of agreement.
4. Conclusions
The present investigation show that we are able to capture the measured raw pressure distributions, lift and pressure and total drag in the attached region of the flow by the CFD 3D tunnel simulations. Additionally, we observe that the increased lift values observed in the 2D tunnel simulations though show agreement with the 2D free simulations. In the present simulations the existence of corner separations and leading edge horseshoe vortices at the wing junction with the tunnel walls may be responsible for canceling the 2D potential tunnel effects. The strength of these 3D effects are strongly dependent on the parameters of the boundary layer approaching the wing. The state of this boundary layer should be further investigated in the future, along with further test of the spanwise loading on the airfoil.

5. Acknowledgement
The work is partially funded by the EUDP project Accelerate and increase quality in aerodynamic and aeroacoustic design loops (AeroLoop), Journal nr.: 64018-0144. The computations are performed on the DTU PC-cluster Sophia.

6. References
[1] Glauert H 1933 Wind tunnel interference on wings, bodies and airscrews Aeronautical research committee reports and memoranda (H.M. Stationery Office) URL https://books.google.dk/books?id=2FQTNAEACAAJ
[2] Allen H J and Vincenti W G 1944 Wall interference in a two-dimensional-flow wind tunnel, with consideration of the effect of compressibility NACA-TR- 782 National Advisory Committee for Aeronautics. Ames Aeronautical Lab.; Moffett Field, CA, United States
[3] Maskell E C 1963
[4] Bussoletti J E, Huffman J R, Krynytzky J A J and Saarist G R 1997 35th Aerospace Sciences Meeting and Exhibit URL www.scopus.com
[5] Klein A, Richter K, Gardner A D, Altmikus A R M, Lutz T and Krmer E 2011 37th European Rotorcraft Forum 2011, ERF 2011 pp 851–859 cited By .8 URL www.scopus.com
[6] Kohzai M, Ueno M, Koga S and Sudani N 2013 Wall and Support Interference Corrections of NASA Common Research Model Wind Tunnel Tests in JAXA (Preprint https://arc.aiaa.org/doi/pdf/10.2514/6.2013-963) URL https://arc.aiaa.org/doi/abs/10.2514/6.2013-963
[7] Grıtskevich M S, Garbaruk A V, Schütze J and Menter F R 2012 Flow, Turbulence and Combustion 88 431–449 ISSN 1573-1987
[8] Menter F R 1993 AIAA paper 1993-2906
[9] Menter F R, Langtry R B, Likki S R, Suzen Y B, Huang P G and Völker S 2004 Proceedings of ASME Turbo Expo 2004, Power for Land, Sea, and Air (Vienna, Austria: ASME) gT2004-53452
[10] Menter F R, Langtry R and Völker S 2006 Flow Turbulence Combust 77 277–303
[11] Leonard B P 1979 Comput. Meths. Appl. Mech. Eng. 19 59–98
[12] Travin A, Shur M, Strelets M and Spalart P 2000 Proceedings of Euromech Coll. 412 LES of Complex transitional and turbulent flows ed Rodi F
[13] Jones B 1936 The Measurement of Profile Drag by the Pitot-traverse Method Aeronautical Research Committee. Reports and memoranda No. 1688 (Cambridge University Aeronetics Laboratory and Aeronautical Research Council (Great Britain) and Air Ministry. Aeronautical Research Committee: H.M. Stationery Office)