

NUMERICAL ANALYSIS OF POWERED TURBOFAN ENGINE FLOW SUPPRESSION EFFECTS

Ch. Wimmer*, M. De Rosa Jacinto da Silva*, M. Berens*

* Technische Universität Wien, Research Group Aircraft Systems E-307-02-2, Lehgasse 6, 1060 Vienna, Austria

Abstract

Future ultra high bypass ratio turbofan engines will improve efficiency by decreasing fan pressure ratios and nozzle pressure ratios. With this shift of the engine's operating point, the fan nozzle is expected to be unchoked even in cruise flight and the fan nozzle flow is affected by the flow conditions behind the nozzle exit. Nacelle external flow and installation effects influence the static pressure behind the engine, leading to a static pressure different from far-field ambient conditions. The interaction between the wing, the nacelle external flow and the engine efflux lead to a variation in the nozzle pressure ratio in unchoked conditions and, therefore, to a change in mass flow rate and gross thrust. The actual pressure at the nozzle is partially disregarded for gas-path measurements based thrust determination in wind-tunnel and flight test data reduction according to industry legacy thrust drag bookkeeping approaches. The consequence is that for a nozzle c_p of 0.2, the net thrust may be overestimated by more than 1% for aircraft equipped with UHBR turbofan engines and fan pressure ratios of 1.2. Berens et al. [1] performed a parametric study using RANS simulations of NASA's Common Research Model (CRM) and a Through-Flow Nacelle (TFN) to investigate how different flight conditions impact pressure coefficients at the engine's nozzle. The c_p results for the NASA HS CRM TFN are particularly high, both in isolated as well as installed conditions, which may have masked some of the airframe induced effects. Nozzle c_p 's for the isolated powered nacelles in [2] are substantially lower compared to the CRM TFN. In this paper, a turbofan propulsion simulator model will be implemented into RANS-simulations with the goal of reducing uncertainties and drawing a more realistic picture of the flow field around the engine both in the isolated as well as the installed configuration. Suppression effects are quantified in terms of the fan nozzle exit pressure coefficient, the applied nozzle pressure ratio and the fan nozzle mass flow. The impact of flow suppression on thrust determination is investigated for a range of cruise flight Mach numbers, various thrust ratings and various angles of attack.

Keywords

Flow Suppression; Turbofan; CFD; Thrust Drag Bookkeeping; CRM; TPS; TFN

NOMENCLATURE

Latin Symbols

$c_{p19,kin}$ pressure coefficient

g acceleration due to gravity m/s^2

p_0 static pressure N/m^2

$q_{0,kin}$ dynamic pressure N/m^2

T_0 static air temperature K

Greek Symbols

α angle of attack deg

Indices

t total

0 ambient

15 fan nozzle inlet

19 fan nozzle exit

Abbreviations

ASC Austrian Scientific Computing

FNPR fan nozzle pressure ratio

FPR fan pressure ratio

HS-CRM high speed common research model

Inst installed configuration

Iso isolated configuration

NEF nacelle external flow

PR19	actual fan nozzle pressure ratio
RANS	Reynolds averaged Navier Stokes
TFN	through flow nacelle
TPS	turbofan propulsion simulator
UHBR	ultra high bypass ratio
WB	wing body
WIN	wing induced

1. INTRODUCTION

The development of modern turbofan engines has led to increasing bypass ratios, lower nozzle pressure ratios, and lower fan pressure ratios, compared to earlier turbofan generations [3]. This led to increased propulsive efficiency, reduced specific fuel consumption, while increasing the diameter of the fan and nacelle, which makes the engine more susceptible to external flow phenomena [4]. For ever larger fans, the additional nacelle drag might eventually surpass the advantage gained from the decrease in fan pressure ratio and the increase in bypass ratio. In this work, ultra high bypass ratio (UHBR) turbofan engines with a bypass ratio of 15 are investigated by means of a turbofan propulsion simulator. Turbofan propulsion simulators (TPS) are considered to provide the most realistic engine flow simulation for turbofan engine-airframe interference tests in wind tunnel tests [5]. A key focus of this work is the use of computational fluid dynamics (CFD) to evaluate installation aerodynamic effects for underwing engine configurations for a modern wide body aircraft [6]. The aerodynamic interaction between the nacelle, the pylon, the wing, and the engine's intake and exhaust streamtubes of modern subsonic transport aircraft creates a variety of complex inter-dependent phenomena [2]. Studying the engine in an isolated configuration and in the installed configuration allows to characterize aerodynamic installation effects qualitatively and quantitatively. Accounting for these effects enables airframe manufacturers to draw a more comprehensive picture of thrust available and also drag in different phases of flight.

The research objectives are formulated to understand flow suppression and its impact on fan nozzle mass flow and gross thrust:

- Quantify the flow suppression effects by means of the fan nozzle back pressure coefficient $c_{p19,kin}$.
- Investigate both the isolated and the installed configuration and outline the installation effects.
- Investigate the effect of flight Mach number on flow suppression.
- Investigate the effect of various thrust ratings on flow suppression.
- Investigate the effects of various angles of attack on flow suppression.

- Quantify the effect on gross thrust depending on the nozzle back pressure for both the isolated and the installed configuration.

2. METHODS

2.1. Turbofan Propulsion Simulator

A 0-D gas dynamics model for a TPS was employed to compute the inputs for representing the operating TPS in CFD simulations. Although there is a fundamental difference in the way a TPS operates compared to an actual turbofan engine, it is believed to represent the flow inside and outside of a turbofan engine/nacelle favourably. A difference lies in the design of the core duct, which is closed off in the front to air that would normally enter through the intake. Instead of a hot combustion gas, compressed air is delivered to the TPS turbine section from an outside source, which is expanded in a turbine that drives the fan. The intake mass flow is directed to the bypass duct (fan duct). Although TPS's are used in wind tunnel testing with scaled-down airframes and engine sizes, simulations were performed with a full-size airframe and a TPS with a fan diameter of $d_{fan} = 3.74m$ and a nacelle length of $6.0m$. Figure 1 shows a cut through the TPS. The numbering system is derived from [7].

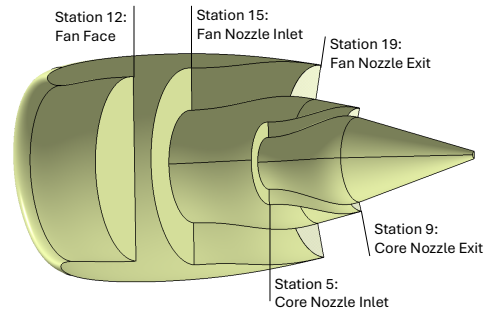


FIG 1. Cut through the CAD model of the TPS with station numbers.

The TPS nacelle assembly consists of the nacelle, the convergent fan and core nozzles and the plug. In contrast to turbofan engines, this CAD model lacks the bifurcation and guide vanes in the fan and core nozzles. The fan spinner is also omitted from the design for simplicity. The fan nozzle is sized to achieve low nozzle pressure ratios typical for UHBR engines to achieve unchoked flow in cruise flight. The high-speed common research model (HS-CRM) [8] is used to investigate installation effects. A pylon was designed in analogy to the CRM's TFN pylon. The installation position and orientation of the TPS model resembles also that of the CRM's TFN.

2.2. Thrust-Drag-Bookkeeping

The question of how much thrust an engine is producing requires a systematic approach to account for

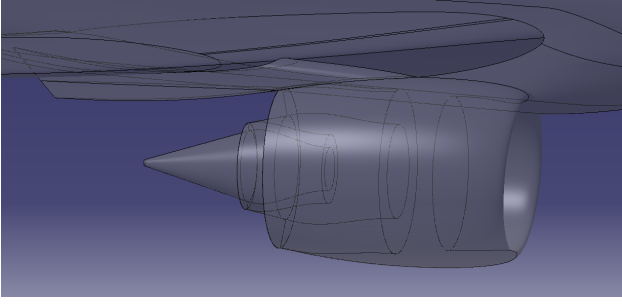


FIG 2. Installed configuration of the TPS model on the HS-CRM wing.

the propulsive and drag forces that act on it. In this work, the main focus will be laid on the bypass duct, i.e. the fan nozzle. The gross thrust produced by the fan nozzle can be bookkept using either the fan nozzle exit station 19 as the measurement station, called the standard gross thrust or a location far downstream of the exhaust where the fan nozzle efflux has expanded to ambient pressure (station 100), called the ideal gross thrust [7], [4], [9].

$$(1) \quad F_{G19} = \dot{m}_{19}v_{19} + A_{19}(p_{19} - p_0)$$

$$(2) \quad F_{G100} = \dot{m}_{19}v_{100}$$

The velocity of the fully expanded jet v_{100} is calculated by assuming isentropic expansion to ambient pressure. The shear and pressure forces that act on the exhaust jet streamtube between stations 19 and 100 are not taken into account in this approach. The ram drag experienced by the engine is defined as

$$(3) \quad F_{RAM} = \dot{m}_{19}v_0$$

The fan nozzle mass flow can be calculated using the thermophysical properties at the fan nozzle exit:

$$(4) \quad \dot{m}_{19} = A_{19} \frac{p_{t19}}{\sqrt{T_{t19}}} \sqrt{\frac{2\gamma}{R(\gamma-1)}}^* PR19^{-\frac{1}{\gamma}} \sqrt{1 - PR19^{\frac{1-\gamma}{\gamma}}}$$

where PR19 is the actual fan nozzle pressure ratio at station 19 (referred to as applied pressure ratio in [9]). In addition, the fan nozzle pressure ratio FNPR is defined, using the ambient static pressure p_0 instead of the local static pressure p_{19} .

$$(5) \quad PR19 = \frac{p_{t19}}{p_{19}}$$

$$(6) \quad FNPR = \frac{p_{t19}}{p_0}$$

The actual fan nozzle pressure ratio for a convergent nozzle is limited, when the flow is accelerated to sonic conditions ($M_{19} = 1$) at the nozzle exit.

$$(7) \quad PR_{crit} = \left(\frac{p_{t1}}{p_i} \right)_{crit} = \left(\frac{\gamma+1}{2} \right)^{\frac{\gamma}{\gamma-1}} = 1.89$$

The velocities v_{19} and v_{100} can either be calculated using the 0-D gas dynamics model or from the CFD-simulation data. The fan pressure ratio

$$(8) \quad FPR = \frac{p_{t15}}{p_{t0}}$$

expresses the increase in total pressure achieved by the fan relative to the ambient total pressure and serves as an important parameter to quantify the thrust rating of the engine. In this work, a design fan pressure ratio of $FPR = 1.3$ is targeted.

2.3. Flow Suppression

The thrust created by a turbofan engine depends on fan and core nozzle performance. High bypass ratio (6) and very high bypass ratio (10) turbofan exhaust systems operate in choked condition in the cruise flight phase [2], which means that the nozzle flows are accelerated to sonic conditions. Therefore, the flow through the nozzle is only governed by the nozzle itself, independent of the conditions downstream of the nozzle. Ultra high bypass ratio (>14) turbofans will operate unchoked, leading to a coupling of the nozzle flow and the flow behind the engine. In flight, two phenomena influence the velocity and the pressure distribution around the back of the engine. One is the flow around the engine nacelle, called the nacelle external flow NEF. The other is the flow field induced by the wing WIN. Both flows interfere with the exhaust jet and affect it by increasing or decreasing the nozzle back pressures relative to ambient conditions. With an increase in fan nozzle back pressure, the actual fan nozzle pressure ratio PR19 decreases, which leads to a decrease in mass flow (flow suppression) and consequently in thrust, when FPR is held constant. Nacelle external and wing flow phenomena are not accounted for when an engine is evaluated in an altitude test facility because of missing nozzle external flow. Hence, the static pressure at the nozzle exit adjusts itself close to the ambient pressure of the flight test case. Flow suppression manifests itself by a non-zero pressure coefficient at the fan nozzle exit.

$$(9) \quad c_{p19,kin} = \frac{p_{19} - p_0}{q_{0,kin}}$$

The static pressure offset is normalized using the compressible definition of the dynamic pressure $q_{0,kin} = p_{t0} - p_0$ to calculate the pressure coefficient. Berens et al. [1], and Campomanes [2] have shown that the static pressure at the fan nozzle exit is higher than ambient pressure due to the nacelle external flow. Under-wing installation is expected to increase the static pressure behind the engine due to the pressure field induced

by the lift-producing wing. Positive values of $c_{p19,kin}$ reduce the actual pressure ratio PR19 and hence the fan nozzle mass flow, relative to the unsuppressed flow ($c_{p19,kin} = 0$). Ultimately, the momentum contribution to the gross thrust is reduced accordingly.

3. PARAMETRIC STUDY SET-UP

The main objective of the parametric study is to establish the governing relations within the TPS and to calculate the relevant geometry and flow parameters which will be used as boundary conditions in the simulation campaign. The parameters in question can be divided into two groups, the geometric parameters of the TPS such as the fan and nozzle cross sections (diameters) and the flow parameters, which change with the free-stream conditions and the thrust rating. Due to the large number of operating conditions analyzed in the simulation campaign, only one engine geometry and one installation location were investigated throughout the study. Two initial engine geometries were reviewed for adequate aerodynamic performance before the third was deemed satisfactory.

3.1. 0-D Gas Dynamics Model

A 0-D gas dynamics model for a TPS was used to calculate the major dimensions of the TPS like the inlet and outlet cross section of the nozzles, the fan diameter and the intake highlight diameter. In addition, the boundary conditions for the CFD-campaign were calculated. The geometric properties of the nozzles and the nacelle were sized to achieve lower nozzle pressure ratios, typical for future UHBR turbofan engines. In a first step, the approximate thrust delivered by the engine was calculated. Assuming a Lift-to-Drag ratio of 20 and a Take Off mass of 254 tons, we can compute the installed propulsive force

$$(10) \quad F_{IPF} = \frac{m_{TO} * g}{\frac{L}{D}} = \frac{254000kg * 9.81m/s^2}{20} \approx 125kN.$$

One engine therefore must deliver about $62.5kN$. Since the gas dynamics model is not able to account for any of the drag contributions, a value of $70kN$ was chosen as reference for sizing the engine in cruise flight conditions ($M_0 = 0.80$ at $FL380$ and ISA conditions). In the next step, the gas dynamics model is employed to compute the boundary conditions to be applied at the fan face (station 12), the fan nozzle inlet (station 15) and the core nozzle inlet (station 5). Figure 1 shows each of the stations.

3.2. Design of Experiment

Turbofan flow suppression is inherently dependent on the pressure and flow fields behind the engine. Additionally, in the installed configuration, where an interaction with the flow field around the wing is expected, the nozzle back pressure is a function of the lift produced by the wing. With respect to the trend

towards higher bypass ratios and lower nozzle pressure ratios, flow suppression is expected to have greater influence on engine mass flow rates with decreasing nozzle pressure ratios and specific thrust. Following this reasoning, the simulation campaign is broken into two branches:

- VTR - Branch: This branch will investigate flow suppression with respect to various thrust ratings (VTR). The angle of attack will be kept constant.
- VA - Branch: This branch will investigate flow suppression with respect to various angles of attack. One representative thrust rating ($FPR \approx 1.3$) will be used.

Withing each of these branches, the Reynolds number will remain the same at roughly $40 * 10^6$ and the flight Mach number will be varied. Each Mach and Reynolds number combination is called a Case.

3.2.1. Thrust Rating Sweeps - VTR

To investigate the suppression effects for different thrust ratings, FPR is varied between 1.00 and 1.35 at constant ambient conditions. A range of four cruise flight Mach numbers were investigated ($M_0 = 0.80, 0.70, 0.55, 0.40$) to outline the Mach number effects. Ambient conditions for each Case correspond to a flight level (FL) in the international standard atmosphere [10]. The thrust dependent properties vary within each Case to investigate different thrust ratings. The tabulated values for ambient conditions and thrust dependent boundary conditions for each Case can be found in [11]. The flight Mach numbers for each case were chosen and the flight altitudes (flight levels) were calculated to maintain a Reynolds number of approximately $40 * 10^6$. Therefore, each Case can be compared using the Reynolds number similarity.

3.2.2. Angles of Attack Sweeps - VA

Investigating the effect of angle of attack on the nozzle back pressure is the second objective of this work. Four thrust ratings, one for each flight Mach number were selected. The angle of attack of the isolated engine and that of the installed configuration was varied between -6° and $+8^\circ$ with increments of $\Delta\alpha = 2^\circ$ for each Case. The thrust related boundary conditions for each Case were kept constant over the angle-of-attack-sweep.

3.2.3. Note on thrust dependent Boundary Conditions

During the simulation campaign, an error was found in the gas dynamics model which lead to a mismatch between the core nozzle and the fan nozzle. Fan nozzle mass flows $\dot{m}_f$ and fan pressure ratios FPR were lower due to the error. Initially, FPR should have been varied between $1.25 < FPR < 1.35$. The flow in the core corresponds to this range in FPR. The simulation results in Cases A ($M_0 = 0.80$) and B

($M_0 = 0.70$) showed, that the actual FPR ranged from $1.10 < FPR < 1.35$. It was decided to keep the data and to extend the simulation campaign to $1.00 < FPR < 1.35$ to investigate thrust ratings and flow suppression that correspond to a fan in idle or through flow condition (FPR=1.0, no power is added to the flow by the fan). The influence of the mismatch between core and fan nozzle flows was investigated for each Case with one data set (isolated and installed configuration) and was found negligible. No upstream influence of the core nozzle exit on the fan nozzle was found. Numerical difficulties arose with the corrected boundary conditions, because at $FPR = 1.0$ the core mass flow decreases to almost zero which led to an oscillating CFD-solution in the core nozzle. This is considered to be a peculiarity of the TPS, which is not supplied with ram air mass flow from the intake like the fan nozzle. The core mass flow of an actual turbofan engine would not decrease to such low values and the assumption of steady state flow would remain true.

4. ENGINE GEOMETRY MODEL

The baseline for every CFD-simulation is the geometry of the problem. The geometry can sometimes be generated in the CFD-application itself. Complex geometries are generated more conveniently with specialized CAD-software. Furthermore, simulation results can indicate the need for improvement of the geometry. If geometric optimization is the main goal of the study, a high fidelity parametric model is needed to allow quick and easy changes of the geometric and hence aerodynamic characteristics. In this section a medium-fidelity parametric model will be developed and the workflow will be laid out. From general aerodynamic properties of aero shapes to a mathematical representation and finally to a point cloud export to the CAD-application [12].

4.1. Parametric Geometry Approach

Following the approach outlined in [13] and [14], a parametric model was set up to generate the geometry of the nacelle external contour, the intake and the exhaust nozzles. The dimensions of the through flow nacelle (TFN) of the high-lift CRM were used as a starting point for the overall geometry. The TFN is approximately 6m long and has a maximum diameter of 4m. A nose droop was incorporated into the nacelle and the intake. Obert et al. [15] describe the various aerodynamic aspects of turbofan engine intakes and exhaust systems which served as the knowledge base in the design of this geometry model. The class shape transformation (CST) curves in this approach are polynomials that provide a simple way to control the most important geometric properties of an airfoil, an engine nacelle external contour, an intake contour or the inner and outer wall contours of an exhaust nozzle. CST-curves are normalized using the fan cowl chord length c or a similar length scale. The x, z -

coordinate system (symmetry plane of the engine) becomes a ψ, ξ -system with

$$(11) \quad \psi = \frac{x}{c} \in [0, 1]. \quad (12) \quad \xi = \frac{z}{c}.$$

The complete dimensionless expression of the desired contour reads as

$$(13) \quad \xi(\psi) = \xi_{le} + C(\psi)S(\psi) + (\xi_{te} - \xi_{le})\psi.$$

with

$$(14) \quad C(\psi) = \psi^{N1}(1 - \psi)^{N2}$$

the class function controlled by the exponents $N1$ and $N2$ and the shape function.

$$(15) \quad S(\psi) = \sum_{i=0}^n bp_i K_{i,n} \psi^i (1 - \psi)^{n-i}.$$

In Equation 13 ξ_{le} and ξ_{te} prescribe leading- and trailing edge offsets from the ψ -axis. ξ_{le} represents, for example, the highlight radius of the nacelle and ξ_{te} the nacelle trailing edge radius. The exponents $N1$ and $N2$ control the behavior of the class function at $\psi = 0, 1$. With $N1 = 0.5$, a vertical tangent at the leading edge can be prescribed. The shape function consists of the binomial coefficient $K_{i,n}$ and the Bernstein coefficients bp_i that are found by solving a system of equations. The order of the shape function n can be changed to satisfy the required number of geometric constraints. A shape function of order n has $n + 1$ Bernstein coefficients bp_i . Since one geometric constraint, for example, the maximum radius of the nacelle affects every Bernstein coefficient, a system of linear equations must be solved. The TPS geometry model consists of four components that were designed using the parametric approach outlined above: The intake contour, the nacelle external contour, the core nozzle and the fan nozzle. For each component, a parameter vector is defined that contains the desired dimensions (maximum nacelle diameter, nose radius, trailing edge diameter, etc.). The model, programmed using the MATLAB [16] Symbolic Toolbox [17] calculates a predefined number of points which are then interpolated in CATIA V5 to produce the contours. To incorporate a nose droop in the nacelle, a second set of parameters for the intake and the nacelle external contour was defined that describes the geometry of the lower nacelle and intake contours. The differences between the upper and the lower contour are restricted to the forward part of the nacelle and are meant to delay boundary layer separation on the lower intake lip at high angles of attack. The parameter vectors $\vec{p}_i(\gamma_i)$ between the upper (0°) and the lower (180°) position are interpolated using a cosine distribution and are a function of the angle of rotation about the nacelle axis γ_i .

$$(16) \quad \vec{p}_i = \vec{p}_u + \frac{\vec{p}_u - \vec{p}_l}{2}(1 - \cos(\gamma_i))$$

This distribution ensures a flat intake lip and a smooth transition from the right to the left half of the nacelle. The results for the 0°, 90° and 180°-contours are shown in Figure 3 below.

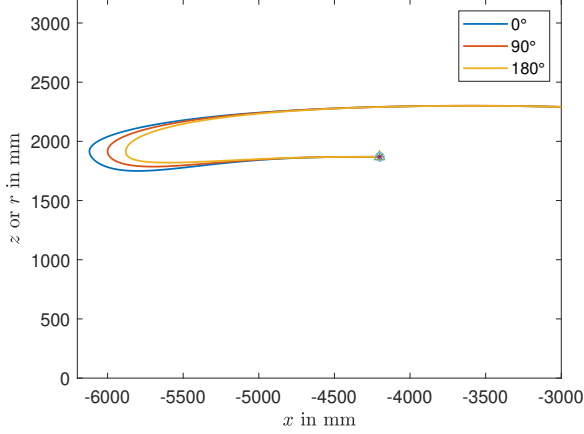


FIG 3. Front half of the nacelle contours and the intake contours at 0°, 90° and 180° of rotation about the engine longitudinal axis. Note the change nose shape and reduction of chord length.

The same process is performed for the exhaust nozzles. The geometry consists of an outer and an inner wall for each nozzle. Instead of using the inner and outer radius of the walls, the decision was made to reformulate the nozzle cross section area such that

$$(17) \quad A(x) = \pi(r_o^2(x) - r_i^2(x)) = 2\pi r_m(x)t(x)$$

with $r_m = (r_o + r_i)/2$ the mean radius and $t = r_o - r_i$ the thickness of the annular channel all depending on the location along the nozzle. Assuming $A(x)$ is known and $r_m(x)$ is a predetermined *CST*-curve, we can express

$$(18) \quad t(x) = \frac{A(x)}{2\pi r_m(x)}$$

and hence

$$(19) \quad r_o(x) = r_m(x) + \frac{t(x)}{2} \quad (20) \quad r_i(x) = r_m(x) - \frac{t(x)}{2}$$

The mean line of the nozzle $r_m(x)$ is calculated using a fifth order polynomial because there is one more constraint to be satisfied. Attention must be paid to the fact that both nozzle exit sections (A_9 , A_{19}) eject the flow not parallel to the nozzle axis (x-direction) but inward towards it. The effective flow areas must be increased proportionally to the angle between the outflow direction and the nozzle axis.

$$(21) \quad A_{19} = \frac{A_{19,eff}}{\cos(\alpha_{exit,19})}$$

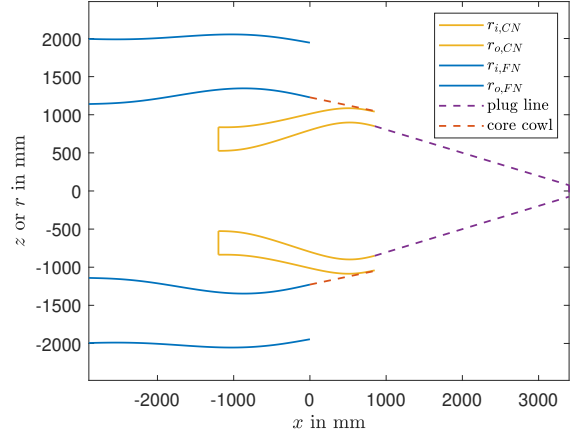


FIG 4. Exhaust nozzle contours with plug line and core cowl. CN and FN denote the core and the fan nozzle.

The angle of the core cowl and the plug line must match the angle of the inner wall at the trailing edge to avoid local shocks. The data is exported to Excel, and finally CATIA is used to generate the surfaces. The inlet and outlet cross sections of both nozzles are large compared to existing engine designs, because they scale with the fan diameter to avoid flow separation on the rear half of the nacelle.

4.2. Installed Configuration

The engine model is now ready for export to Star-CCM+ and to perform all simulations for the isolated configuration. For the installed configuration, a pylon is created to connect the outside of the nacelle and the wing. This is done in the same manner as for the nacelle and the exhaust nozzles, using *CST*-curves. The bifurcation, which would protrude into the fan nozzle, was omitted to simplify the design. The spanwise installation location of the TPS engine model is identical to that of the CRM-through flow nacelle (TFN), 1.5m inboard of the Yehudi break. The installation angles are defined relative to the wing-body-reference system (x: backward against direction of flight, z: upward). The engine is rotated +4.5° about its longitudinal axis, +2.86° upward and +2° inward. The installation location and orientation of the TPS were derived from those of the TFN. The nacelle is 6m long and its trailing edge is positioned below the leading edge of the wing where it intersects the pylon. The installed configuration, taken from CATIA is shown in Figure 2.

4.3. Simulation Feedback

Although the objective of this work is not aerodynamic optimization, the parametric approach was chosen to allow quick and easy changes of the engine geometry.

Initial simulation results were used to highlight aerodynamic deficiencies and facilitated the improvement of the nacelle and the exhaust geometries. All figures in this work show the final version of the engine geometry.

5. COMPUTATIONAL APPROACH

The investigation of turbofan flow suppression requires sophisticated wind tunnel tests or an adequate computational fluid dynamics (CFD) approach. Although current thrust-drag-bookkeeping approaches [4], [7] still do not require information about the static pressure in the nozzle exit sections in wind tunnel testing, de Wolf [5] outlined their importance and advocated its measurement. CFD simulations can be used complementarily to wind tunnel testing, but also as a standalone tool to investigate turbofan and TPS performance and airframe integration. In this chapter, the simulation setup is explained and the results of the convergence study are presented.

5.1. The ASC - Austrian Scientific Computing

The research objectives formulated above require full three-dimensional CFD simulations of turbulent viscous flow for both the isolated and the installed configuration. The investigation of various flight Mach numbers, thrust ratings and angles of attack increase the required computational resources for this work. Computations were performed on the Austrian Scientific Computing infrastructure, ASC (formerly known as Vienna Scientific Cluster or VSC). Both the older VSC-4 and the newer VSC-5 partitions were employed.

5.2. Siemens Simcenter Star-CCM+

The availability of a preexisting simulation setup [1] in Siemens' Simcenter Star-CCM+ [18] advocated the choice for this software. Works of a similar scope such as [19] served as a guideline.

5.3. Numerical Setup

At the beginning of each CFD study, a number of assumptions have to be made with respect to the nature of the flow field. Time-independent governing equations not only reduce the numerical expenses but also reduce the parameter envelope under investigation to a range where only steady-state flow occurs. For the ranges of flight Mach numbers, thrust ratings and angles of attack studied in this work, a steady-state solution is assumed a priori. Three-dimensional steady-state fully-turbulent Reynolds Averaged Navier Stokes equations (RANS) together with the Spalart-Allmaras turbulence model [20] and an all y^+ wall treatment model were selected. The boundary layer was assumed to be fully turbulent. The Spalart-Allmaras turbulence model is a one-equation model that solves a transport equation for the modified diffusivity to determine the turbulent eddy viscosity [21]. The all y^+ wall treatment was

suggested by the documentation [21] and because of the presence of both external flow about the wing, the fuselage and the nacelle as well as internal flow in the intake and the exhaust nozzles. A coupled solver for the mass, momentum, and energy conservation equations was employed and the ideal gas equation was used. All no-slip walls were assumed adiabatic. The coupled implicit solver employed a 2nd-order upwind discretization scheme.

5.4. Mesh

The domain under investigation is divided into two subdomains, following initial numerical difficulties with the boundary condition at the fan face: the domain inside the engine intake and the exterior domain, consisting of the continuum around the engine and both nozzles. For the exterior domain, Star-CCM+'s tetrahedral surface mesher was employed and an unstructured polyhedral mesher filled the core volume while the volume close to the no-slip walls was discretized using a prism layer mesher. The surface mesh on the interface between the exterior and the intake domain was projected onto the fan face along the circumference of the intake using the directed meshing operation. With this approach, both the prism layer cells and the core volume cells are projected into the intake and create conformal mesh. It was decided to omit the spinner from the engine geometry to simplify the directed meshing operation in the intake region. The meshing operation for the installed configuration followed the same process as in the isolated configuration with the addition of the pylon, the wing and the fuselage in the installed configuration.

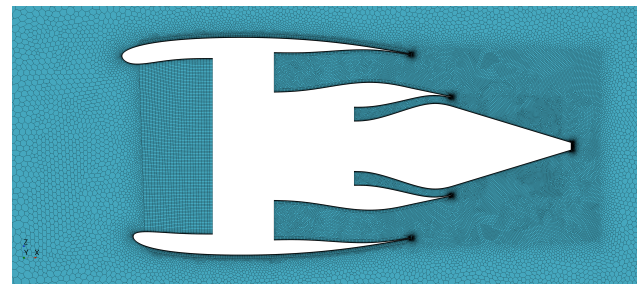


FIG 5. Mesh in the symmetry plane of the engine. Note the conformal transition in the intake. The mesh in this figure shows a coarser mesh than that used in the final simulation.

5.5. Boundary Conditions and Symmetries

The boundary conditions discussed above were implemented. While the fan and the core nozzle inlets were prescribed with the respective mass flow entering the fluid domain at a defined total temperature using a Mass Flow Inlet boundary condition, the fan face was prescribed with an Outlet boundary condition, solely specifying the mass flow that leaves the intake domain. The symmetries of the isolated and installed configura-

rations were exploited to reduce the computational expenses. For the isolated configuration, this also meant that the mass flow rates through the engine have to be multiplied by a factor of 0.5. With appropriate outlet guide vane designs, the residual flow swirls can be practically reduced to zero for fan as well as core flows. Hence, flow swirl is not modeled. Figure 6 shows the boundary conditions as visualized in Star-CCM+. The external boundaries consist of the symmetry plane and a half-cylinder. The front half-circle and the circumference of the half-cylinder are prescribed with the far-field boundary condition, static temperature T_0 , static density ρ_0 , Mach number M_0 and velocity vector $\vec{v}$ as function of the angle of attack α . The back half-circle of the cylinder is defined as a Pressure Outlet. In the isolated configuration, the half-cylinder is 170m long and has a diameter of 80m. The engine is placed and aligned with the center axis of the cylinder 50m behind the front of the domain.

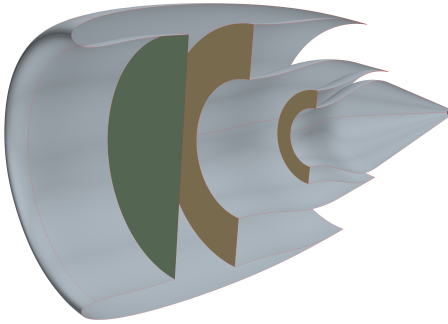


FIG 6. Internal boundary conditions of the engine. The Outlet boundary condition (green) at the fan face and the Mass Flow Inlet boundary conditions (light brown) at the nozzle inlets are used to model the thrust production and the corresponding mass flows through the engine. Grey surfaces are the no-slip adiabatic walls.

The installed configuration uses the symmetry of the whole airframe. Inhere, the half-cylinder is 980m long and has a diameter of 400m. The airframe is placed and aligned with the center axis of the cylinder 200m behind the front of the domain.

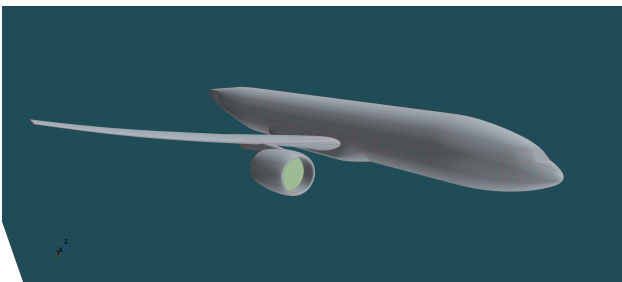


FIG 7. Placement of the symmetry plane for the installed configuration, employing the HS-CRM.

5.6. Data Reduction

CFD simulations create large amounts of data that have to be reduced to extract the most important physical quantities. Area averaged physical quantities at the most important reference stations of the TPS were evaluated according to Equation 22.

$$(22) \quad \bar{q} = \frac{1}{A} \int_A q dA = \frac{\sum_i^n q_i A_i}{\sum_i^n A_i}$$

q is some flow property evaluated in the discretized reference area A (that is, the fan nozzle exit area). In the mesh, the reference area is decomposed into a number of n surface elements A_i with a respective value of the quantity q_i .

5.7. Mesh Convergence Study

When a steady-state CFD simulation is conducted, the simulation is considered to be converged when the residuals have decayed by several orders of magnitude and the solution does not change for a sufficient number of iterations. The converged solution depends on the resolution of the grid in the domain. Both configurations were meshed with different degrees of refinement, starting from a coarse grid and continuously refining it. Five different meshes for the isolated configuration and eight different meshes for the installed configuration were created. The cell sizes in each mesh were parametrically controlled via a base length that was continuously decreased to create refined meshes. Figure 8 shows the increase in the number of cells with each refinement step. To save computational resources in the simulation campaign, a trade-off has to be made between numerical accuracy and computational cost.

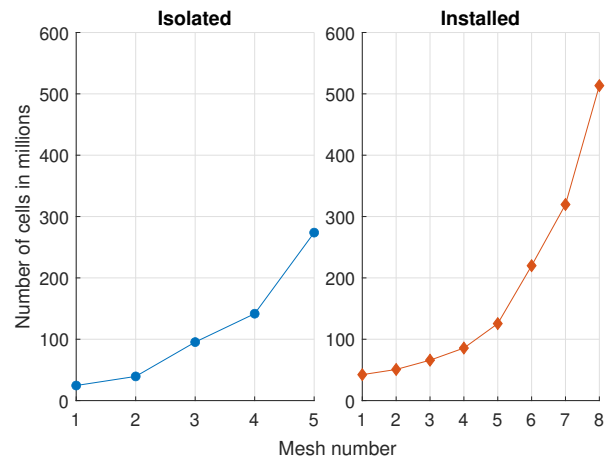


FIG 8. Number of cells for both configurations at each refinement step. With respect to the residuals, convergence was achieved for all meshes.

Each physical quantity q that was monitored was plotted with its relative value q_{rel} . The relative value is defined as the quantity in a refinement step q_i divided by its value in the highest refinement step q_n of the respective configuration.

$$(23) \quad q_{i,rel} = \frac{q_i}{q_n}$$

The normalized quantity highlights the change with increasing mesh refinement while the absolute quantity gives a first impression about the solution itself. The nozzle exit stations were chosen for the mesh convergence study because of the relevance of these stations for this work. The solution on a certain mesh i was considered converged for $|q_{i,rel} - q_{n,rel}| \leq 0.02$.

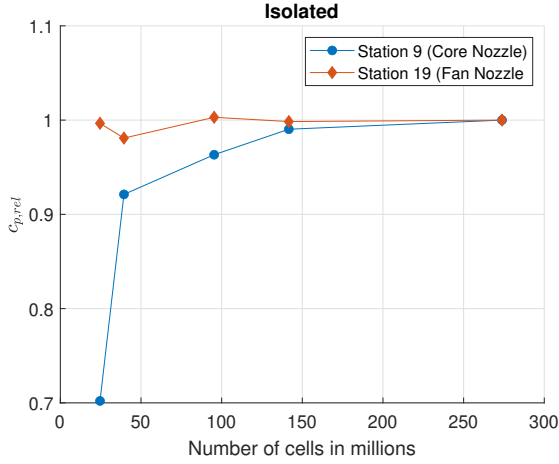


FIG 9. Convergence behavior of the (relative) nozzle exit pressure coefficients for the meshes in the isolated configuration.

Figure 9 shows the relative pressure coefficient at the core nozzle exit (station 9) and the fan nozzle exit (station 19) with an increasing number of cells. $c_{p9,rel}$ increases monotonically and exhibits a large relative change from coarse to fine, and $c_{p19,rel}$ remains relatively constant. $c_{p19,rel}$ satisfies the mesh convergence criteria for all refinement steps, while $c_{p9,kin}$ shows a deviation slightly below 5% at roughly 100 million cells. This deviation is deemed acceptable since the main focus is laid on station 19. For these reasons, the mesh with approximately 95 million cells was chosen for the isolated configuration and the entire simulation campaign.

In the installed configuration, the relative pressure coefficients $c_{p9,rel}$ and $c_{p19,rel}$ in Figure 10 show convergence for the mesh with just over 200 million cells. Although the relative fan nozzle exit pressure coefficient does not change much with mesh refinement, the relative core nozzle exit pressure coefficient exhibits the same behavior as in the isolated configuration. Following these results, the mesh with about 230 million cells was chosen for the simulation campaign.

5.8. Validation: AIAA Drag Prediction Workshop IV

The installed configuration was further investigated. Using data available in [22] from Drag Prediction Workshop IV [22], the lift and drag coefficients of the wing-body combination (WB) from DPW IV and the

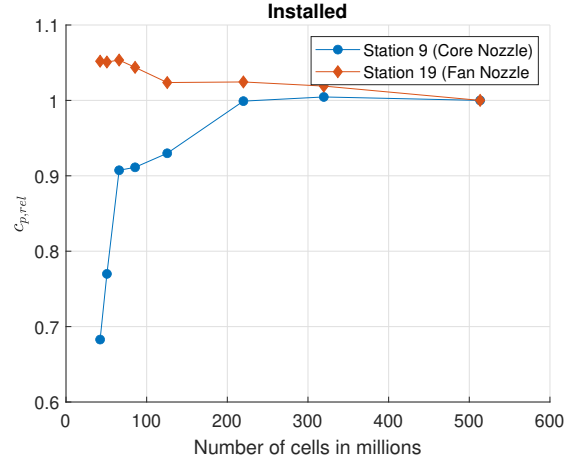


FIG 10. Convergence behavior of the (relative) nozzle exit pressure coefficients for the meshes in the installed configuration.

simulation campaign were compared in addition to the preceding mesh convergence study. DPW IV data is available for the no-tail configuration and for the same Reynolds number ($\approx 40 \cdot 10^6$) and $M_0 = 0.85$. The lift and drag coefficients of DPW IV were defined using $q_0 = 0.5p_0\gamma M_0^2$ and were corrected to $q_{0,kin}$ using Equation 24 to ensure comparability with the simulation data in this work. The relation can be used for pressure, lift and drag coefficients.

$$(24) \quad c_{p,kin} = \frac{\frac{1}{2}\gamma M_0^2}{\left(1 + \frac{(\gamma-1)}{2}M_0^2\right)^{\frac{\gamma}{\gamma-1}} - 1} c_p$$

Figure 11 shows the lift coefficient in the investigated range of angles of attack and Mach numbers. The lift is in good agreement with the data from DPW IV in the published range from 0° to $+4^\circ$ angle of attack. At high angles of attack, the lift decays with increasing flight Mach number. This is expected as a result of the formation of strong shocks on the top side of the wing. Figure 12 shows the drag curve. The simulations underestimate the drag on the wing-body relative to the results of DPW IV. The lift-to-drag ratio is overestimated as a result.

6. COMPUTATIONAL RESULTS

The results of this study are split into two parts. The results of the VTR-study and those of the VA-study. A visual representation of the flow field is presented and the data discussed. In both parts, flow suppression is investigated, and the effects of the installation are described. Finally, implications for thrust-drag-bookkeeping are explained.

6.1. Thrust Variation Study - VTR

This section introduces the results of the thrust variation study for the isolated and the installed configura-

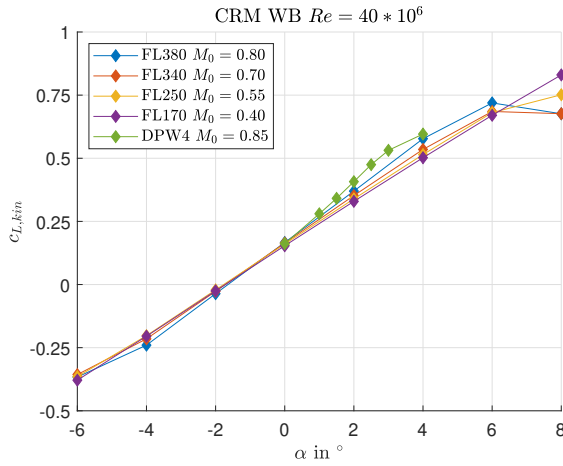


FIG 11. Lift coefficient of the wing-body (WB). Simulation data in comparison with the results of DPW IV.

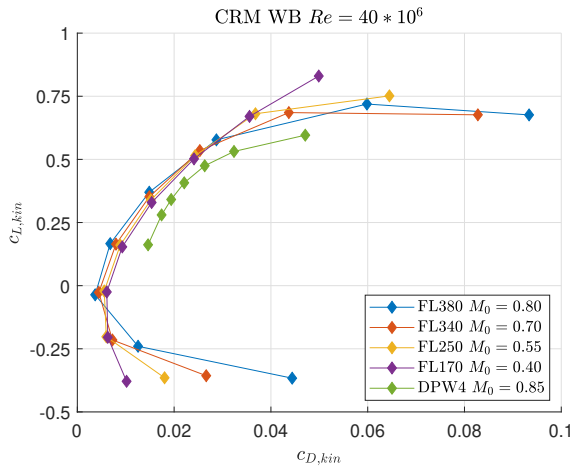


FIG 12. Drag curve of the wing-body (WB). Simulation data in comparison with the results of DPW IV.

ration, describes the installation effects and their dependence on the thrust rating. Chapter 6.1.1 gives a visual impression of the flow in the engine symmetry plane for various thrust ratings. In Chapter 6.1.2 the results are plotted over the fan pressure ratio, highlighting the relation between flow suppression and the thrust generated by the fan.

6.1.1. Flow Visualizations - VTR

Contour plots of the Mach number distribution were generated to visualize the flow in the symmetry plane of the TPS. The selected thrust ratings represent an idle thrust rating with $FPR = 1.0$, the design thrust rating with $FPR = 1.3$ and the highest simulated thrust rating of $FPR = 1.35$. The plots highlight the changes in the flow pattern with changing flight Mach number. Figures 13 and 14 show the isolated and the installed configuration at $\alpha = 0^\circ$, respectively. The TPS in the isolated configuration is oriented normal to the oncoming flow while in the installed configuration it is oriented at an installation

angle of $\Delta\alpha_{TPS} \approx 2.86^\circ$. At $M_0 = 0.80$, the isolated configuration exhibits only slight asymmetries between the top and the bottom side of the nacelle due to the nose droop of the intake and the different contours of the intake lips. The outside of the lower intake lip experiences a weak shock. The exhaust flow is almost symmetric. The installed configuration experiences a shock on the outside of the upper intake lip and an asymmetry between the top and bottom fan nozzle exhaust stream is present. The presence of the wing and the pylon lead to strong shocks behind the fan nozzle near the trailing edge of the core cowl. The convex curvature of the bottom side of the pylon increases the Mach number in the fan exhaust jet below the pylon relative to the bottom side. It must be noted here, that the super-velocities beneath the pylon are the result of the missing bifurcation. With decreasing flight Mach numbers, the shocks around the intake and the core cowl trailing edge fade. In all installed configurations, the exhaust jet is deflected downward, which results from the installation angle of the TPS and the downwash from the wing. Flow visualizations for the other flight Mach numbers investigated can be found in [11], for reference.

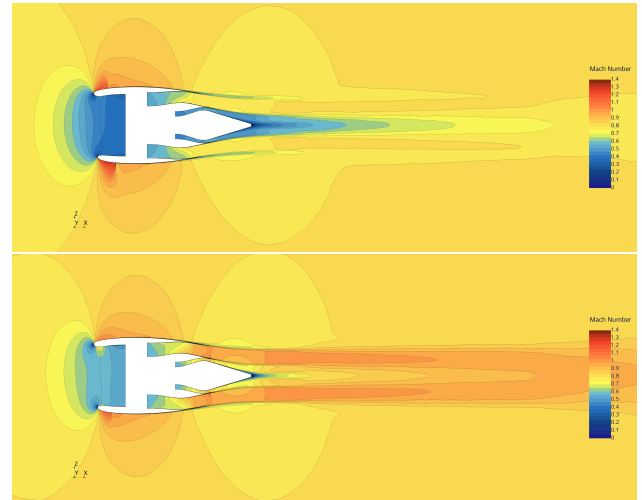


FIG 13. Contour plot of the Mach number in the engine symmetry plane of the isolated configuration for $FPR \approx 1.00, 1.30$.

6.1.2. FPR-related Results

The fan pressure ratio is a quantity that reflects the thrust rating of the TPS. Figure 15 shows the change of $c_{p19,kin}$ with the fan pressure ratio. The latter ranges from $1 < FPR < 1.4$ from the lowest to the highest thrust rating. In all flight cases, the pressure coefficient at the fan nozzle exit increases with FPR. Each blue circle represents a data point for the isolated configuration and each red diamond represents one for the installed configuration. The corresponding data points use the same boundary conditions. The installation effect manifests itself in the offset of the red curve from the blue curve. Installation reduces $c_{p19,kin}$ in most flight cases. FPR also

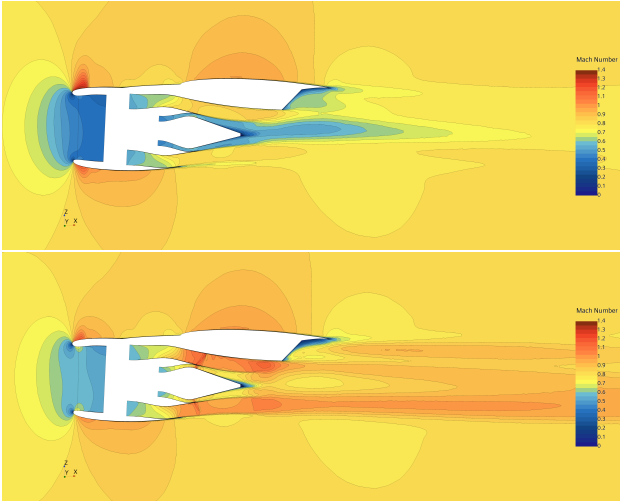


FIG 14. Contour plot of the Mach number in the engine symmetry plane of the installed configuration for $FPR \approx 1.00, 1.30$.

decreases slightly with the installation on the airframe. These results lead to the conclusion, that installation reduces flow suppression significantly at an angle of attack $\alpha = 0^\circ$ for all flight Mach numbers and most thrust ratings investigated. With increasing FPR, the installation effect on $c_{p19,kin}$ decreases and vanishes for $M_0 = 0.80$ and $FPR \approx 1.35$. At this operating point, the Mach number at the fan nozzle exit M_{19} becomes close to unity [11], and no upstream transport of information about the ambient pressure behind the engine is possible. Flow suppression is present in all cases ($c_{p19,kin} > 0$) but the installation effect diminishes towards higher fan pressure ratios.

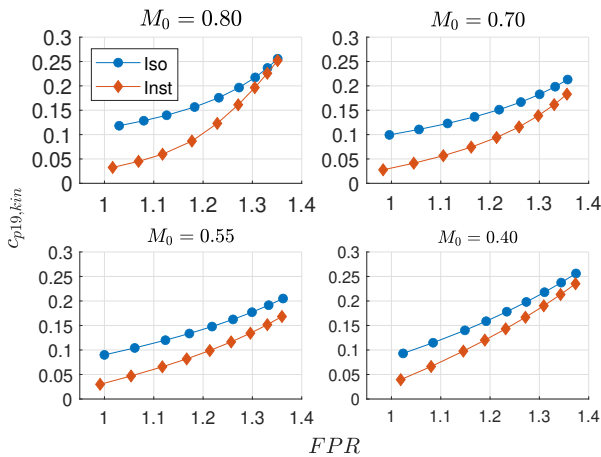


FIG 15. Fan nozzle exit pressure coefficient $c_{p19,kin}$ over a range of fan pressure ratios and different flight Mach numbers.

Figure 16 highlights the difference between FNPR and PR19 for all flight Mach numbers and thrust ratings. For increasing flight Mach numbers and thrust ratings, FNPR shows an over-proportional increase relative to PR19. The pressure ratios increase with the flight

Mach number. When PR19 approaches the critical value for the convergent nozzle, the installation effect fades again. At $M_0 = 0.80$ and $PR19 \approx 1.70$ FNPR starts to deviate strongly from the unsuppressed state ($FNPR = PR19$). FNPR reaches values close to 2.1.

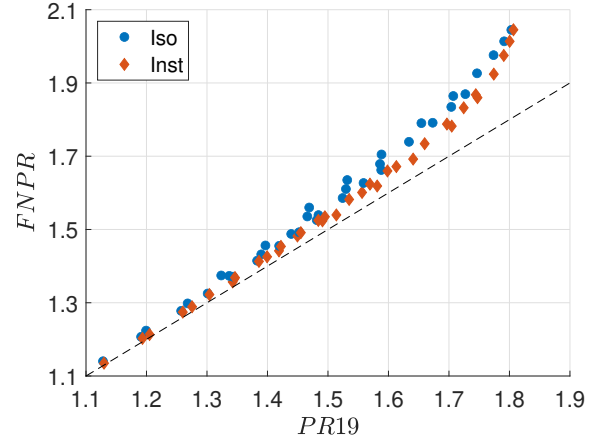


FIG 16. The fan nozzle pressure ratio FNPR plotted over the nozzle's actual pressure ratio PR19. The dashed black line represents $FNPR = PR19$. The data points in this plot come from all flight Mach number $0.40 < M_0 < 0.80$.

The actual pressure ratio PR19 increases from 1.1 to 1.81 with the flight Mach number and thrust rating. For the highest flight Mach number, PR19 seems to converge towards a peak value below the critical pressure ratio. This likely follows from the interaction of the nacelle external flow with the fan nozzle efflux downstream of station 19, which displaces the location, where the fan nozzle efflux reaches the highest Mach number and pressure ratio, to a location downstream of the geometric fan nozzle exit (station 19).

6.1.3. Interim Summary - VTR

The shortcomings of FNPR as a measure for the operating point of the fan nozzle with respect to PR19 were highlighted. The TPS experiences much lower fan nozzle pressure coefficients than the CRM's TFN. Berens et al. [1] reported $c_{p19} = 0.40$ without and $c_{p19} = 0.32$ with a core body in the installed configuration. The results in this work highlight the impact of the nacelle and exhaust geometry as well as the operating point of the fan nozzle on the nozzle back pressure and its sensitivity to the installation effects.

6.2. Computational Results - VA

This section introduces the results of the angle-of-attack variation study for the isolated and the installed configuration, describes the installation effects and their dependence on the angle of attack. Section 6.2.2 will give a description of the flow field in the engine symmetry plane. Before continuing with the results in Section 6.2.3, the definitions of the angle

of attack for both configurations is reviewed and the comparability of both definitions discussed.

6.2.1. Angle of Attack Definition

In the isolated configuration, the angle of attack α_{TPS} is defined as the angle between the TPS's longitudinal or x-axis and the direction of the oncoming flow which was defined using the far-field boundary condition. In the installed configuration, the fuselage longitudinal axis is used as a reference to define α_{WB} . The installation angle $\Delta\alpha_{TPS}$ of the TPS, described in Section 4.2 together with the flow angle induced by the wing $\Delta\alpha_{WIN}$ influence the angle of attack that the TPS experiences in this configuration. While the installation angle is a fixed quantity, the flow angle induced by the lift-producing wing is variable and lift-dependent.

$$(25) \quad \alpha_{TPS} = \alpha_{WB} + \Delta\alpha_{TPS} + \Delta\alpha_{WIN}$$

Moreover, the incidence of the flow field induced by the wing varies such that the highlight area experiences a different effective angle of attack compared to the exhaust nozzles. It was not attempted to match the respective local angles of attack. For these reasons, the direct comparison of the geometrically defined angles of attack α_{TPS} and α_{WB} was prioritized to achieve a robust representation of the angle of attack in the following sections. The subscripts TPS and WB will be omitted from here on and only α will be used.

6.2.2. Flow Visualizations - VA

In Figure 17 the effects of the angle of attack are visualized exemplarily at $M_0 = 0.80$ for the installed configuration from $-2^\circ < \alpha < +4^\circ$. The increasing angle of attack reduces the velocities in and behind the fan nozzle. With the increase in α the shocks form and vanish again around the upper and lower intake lip, respectively. The shock behind the fan nozzle underneath the pylon at $\alpha = -2^\circ$ weakens with increasing angle of attack. At $\alpha = +4^\circ$, a strong shock is present on the upper intake lip. Note that any white areas in the fluid domain represent regions, where the Mach number exceeds the bounds of the colorbar $0 < M < 1.4$. In [11], the Mach number- and $c_{p,kin}$ -plots are presented for all flight Mach numbers and both configurations.

6.2.3. Results

For each flight Mach number, the design thrust rating $FPR \approx 1.3$ was investigated under various angles of attack. The following figures contain the results that allow quantification of the flow suppression and the installation effect. Figure 18 shows the relation between $c_{p19,kin}$ and α from -6° to $+8^\circ$. The isolated configuration shows a maximum at $\alpha = 0^\circ$ at all flight Mach numbers that were investigated. Both decreasing and increasing angles of attack lead to a slight reduction in the pressure coefficient. The initial decrease of $c_{p19,kin}$ for $0.80 > M_0 > 0.55$ and subsequent increase for

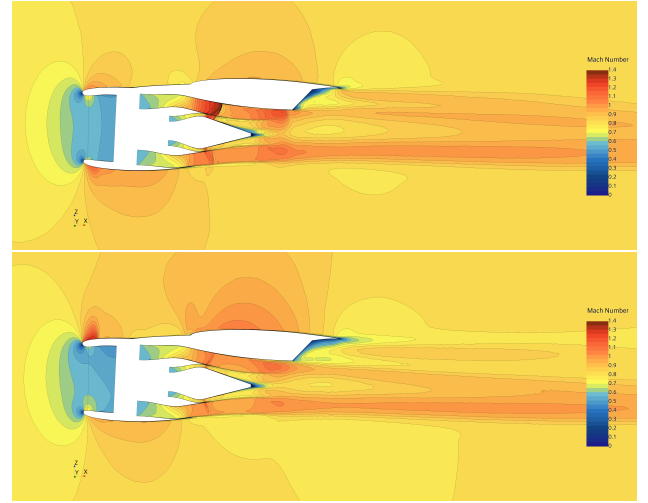


FIG 17. Contour plot of the Mach number in the engine symmetry plane of the installed configuration for $FPR \approx 1.30$ and $\alpha = -2^\circ$ (top) and $\alpha = +2^\circ$ (bottom).

$M_0 = 0.40$ is also found in the results of the VTR study in Chapter 6.1. The installed configuration shows a much stronger dependence on the angle of attack, as the pressure coefficient increases with it. The gradient $dc_{p19,kin}/d\alpha$ decreases with the flight Mach number. The pressure coefficients at the fan nozzle exit are identical for the isolated and the installed configuration at $\alpha = +1^\circ$ at $M_0 = 0.40$, increasing to $\alpha = +2^\circ$ at $M_0 = 0.80$. Outside of these bounds, highly negative angles of attack reduce $c_{p19,kin}$ and even negative values can be achieved. For angles of attack greater than $+2^\circ$ the pressure coefficient of the installed configuration exceeds that of the isolated configuration. These results are expected because the pressure field induced by the wing leads to an increase in static pressure on the bottom side of the wing when the angle of attack is increased further.

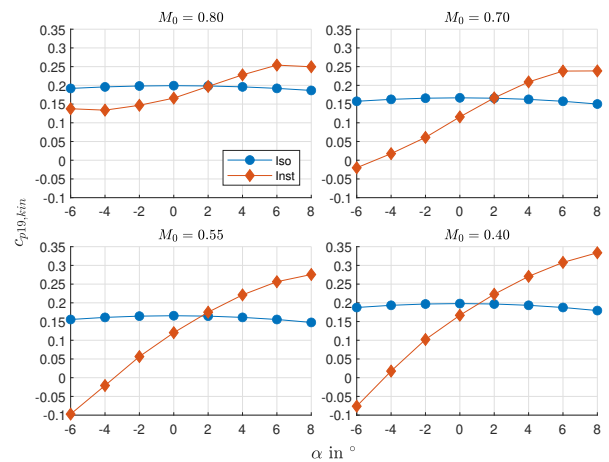


FIG 18. Fan nozzle exit pressure coefficient $c_{p19,kin}$ at $FPR \approx 1.30$ and different flight Mach numbers.

Finally, the fan pressure ratio is shown in Figure 19. FPR varies with the angle of attack in the order of 1% – 2% for the installed configuration. The isolated configuration shows even less dependency on α . In all flight cases, the fan pressure ratio is found slightly below the targeted value of 1.3.

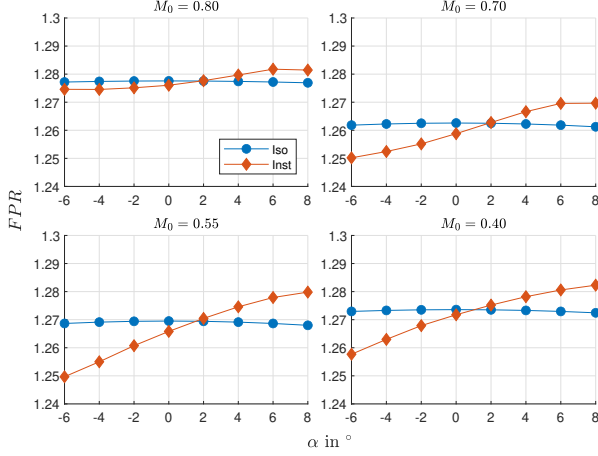


FIG 19. Fan pressure ratio FPR for various angles of attack and flight Mach numbers.

6.2.4. Interim Summary - VA

Flow suppression and installation effects have been discussed from the viewpoint of the angle of attack. Shortcomings that arose from the simulation strategy have been discussed and their impact on the usability of the data was outlined. It was shown that sufficiently negative angles of attack can produce a negative suppression effect in the installed configuration and negative pressure coefficients. For angles of attack higher than 1° to 2°, $c_{p19,kin}$ rises above the value for the isolated configuration.

6.3. Implications to Engine Performance

In this section, the implications for TPS performance are derived from calculations with the 0-D gas dynamics model. The simulation results for $c_{p19,kin}$ together with the respective thrust ratings and flight cases are used to prescribe the static pressure at the nozzle exit and compare the suppressed ($c_{p19,kin} \neq 0$) with the corresponding unsuppressed ($c_{p19,kin} = 0$) flow. The model is then used to compute the fan mass flow $\dot{m}_f$ and the ideal net thrust F_{N100} produced by the bypass duct, defined as

$$(26) \quad F_{N19} = \dot{m}_f(v_{19} - v_0) + A_{19}(p_{19} - p_0).$$

Equation 26 is not defined in accordance with [4] because the gas dynamics model cannot compute the drag contribution of the fan nozzle efflux impinging on the core cowl afterbody. For each flight case (thrust rating, α , M_0), the suppressed (sup) and unsuppressed

(unsup) data are calculated to compute the relative deviations ε (defects).

$$(27) \quad \varepsilon_{\dot{m}_f} = \frac{\dot{m}_f(c_{p19,kin} \neq 0) - \dot{m}_f(c_{p19,kin} = 0)}{\dot{m}_f(c_{p19,kin} = 0)} = \frac{\dot{m}_{f,sup} - \dot{m}_{f,unsup}}{\dot{m}_{f,unsup}}$$

$$(28) \quad \varepsilon_{F_{N19}} = \frac{F_{N19}(c_{p19,kin} \neq 0) - F_{N19}(c_{p19,kin} = 0)}{F_{N19}(c_{p19,kin} = 0)} = \frac{F_{N19,sup} - F_{N19,unsup}}{F_{N19,unsup}}$$

Following this definition, the suppressed nozzle flow experiences PR19 at the nozzle exit, while the unsuppressed nozzle flow experiences FNPR. Both quantities are computed, visualized and discussed. Figure 20 shows the results for the fan mass flow and the net thrust defect using the data for $c_{p19,kin}$ of the VTR-study, where the thrust rating (FPR) was varied. The mass flow defect decreases with the flight Mach number. The highest mass flow defects in the isolated configuration occur for the lowest thrust ratings. They reach between -2% and -3%. The installation and hence negative suppression significantly reduces the mass flow defect to between -1% and 0%, because the reduction in nozzle back pressure due to installation increases PR19 and hence the mass flow. This installation behavior is deemed characteristic for a negative installation effect (reducing $c_{p19,kin}$). In each subfigure of Figure 20, a dashed black line is drawn through an arbitrary pair of corresponding isolated and installed data points. For both data points, the thrust rating is the same and only $c_{p19,kin}$ prescribed at the nozzle exit is different. All four dashed black lines roughly intersect the origin ($\varepsilon_{\dot{m}_f} = 0$, $c_{p19,kin} = 0$) indicating that the mass flow defect diminishes for $c_{p19,kin} = 0$. For increasing pressure coefficients, the mass flow defect increases (negative slope of the dashed black line). Note that the linear relation between $\varepsilon_{\dot{m}_f}$ and $c_{p19,kin}$ indicated by the dashed black line is an approximation. The mass flow defect due to flow suppression does not create a net thrust defect. The net thrust defect is almost constant for medium to high thrust ratings and high pressure coefficients. At low thrust ratings, the net thrust defect reaches up to +8% at $M_0 = 0.70$ due to the non-zero pressure term in the net thrust of Equation 26. For the unsuppressed flow, this term is always zero. At these thrust ratings, the fan pressure ratio is already very low. The installation effect shifts the installed curve to lower pressure coefficients. It must be noted here, that the first data set in each flight case was omitted because FPR decreases below 1 and the net thrust defects jump to negative values. This operating point is not of interest in this section. The increase in net thrust due to flow suppression is a peculiarity that follows from the use of station 19 for thrust bookkeeping and the definition of F_{N19} in Equa-

tion 26. Flow suppression decreases $\dot{m}_f$ and since the power provided to the fan is kept constant (the turbine is believed unaffected by flow suppression), the excess power is converted into higher total pressure p_{t15} and hence FPR. The operating point of the fan is shifted towards a lower mass flow rate but a higher fan pressure ratio. At lower thrust ratings (low FPR), flow suppression reduces $v_{19,sup}$ compared to $v_{19,unsup}$, reducing the momentum contribution to $F_{N19,sup}$. At the same instance, the pressure contribution of $p_{19,sup}$ to the net thrust remains constant (because it is prescribed with $c_{p19,kin}$), leading to a higher net thrust under flow suppression. In the VTR-study, the pressure coefficient was shown to have a positive non-zero value at $FPR = 1$ and hence, the pressure thrust contribution $A_{19}(p_{19} - p_0)$ remains positive. When FPR approaches unity, there must not be produced any installed propulsive force by the TPS. Spillage drag acting on the intake stream tube and the post-exit drag acting on the expanding exhaust efflux must increase to compensate, in part, for the pressure contribution to F_{N19} . Airframe drag also increases because the nacelle drag increases as a result of the increased spillage. Figure 21 shows the same results using the $c_{p19,kin}$ -data from the VA-study, where the angle of attack was varied and the thrust rating (i.e. the CFD-boundary conditions) was kept constant. The installed configuration shows large variations in pressure coefficient and hence mass flow. With increasing angles of attack the mass flow defect becomes as low as -2% . For highly negative angles of attack, the mass flow defect can become positive up to $+0.5\%$. The net thrust defect does not exceed $+0.25\%$ at $FPR \approx 1.3$. It shows a weak dependency on the pressure coefficient. While the net thrust defect is slightly positive for $0.80 < M_0 < 0.55$, it is slightly negative for $M_0 = 0.40$ and $c_{p19,kin} > 0$. The mass flow defects exhibit a slight non-linear dependency on $c_{p19,kin}$ approaching the unsuppressed state, compared to the black dashed lines in Figure 20. These results highlight the impact of varying nozzle back pressures on an engine that operates at its design $FPR = 1.3$. The net thrust defect, as defined in Equation 26 does not exceed $\pm 0.25\%$.

Figure 21 shows the same results using the $c_{p19,kin}$ -data from the VA-study, where the angle of attack was varied and the thrust rating (i.e. the CFD-boundary conditions) was kept constant. The installed configuration shows large variations in pressure coefficient and hence mass flow. With increasing angles of attack the mass flow defect becomes as low as -2% . For highly negative angles of attack, the mass flow defect can become positive up to $+0.5\%$. The net thrust defect does not exceed $+0.25\%$ at $FPR \approx 1.3$. It shows a weak dependency on the pressure coefficient. While the net thrust defect is slightly positive for $0.80 < M_0 < 0.55$, it is slightly negative for $M_0 = 0.40$ and $c_{p19,kin} > 0$. The mass flow defects exhibit a slight non-linear dependency on $c_{p19,kin}$ approaching the unsuppressed state, compared to the black dashed lines in Figure 20. From these calculations the conclusion can be drawn, that flow suppression leads to a reduction in fan mass

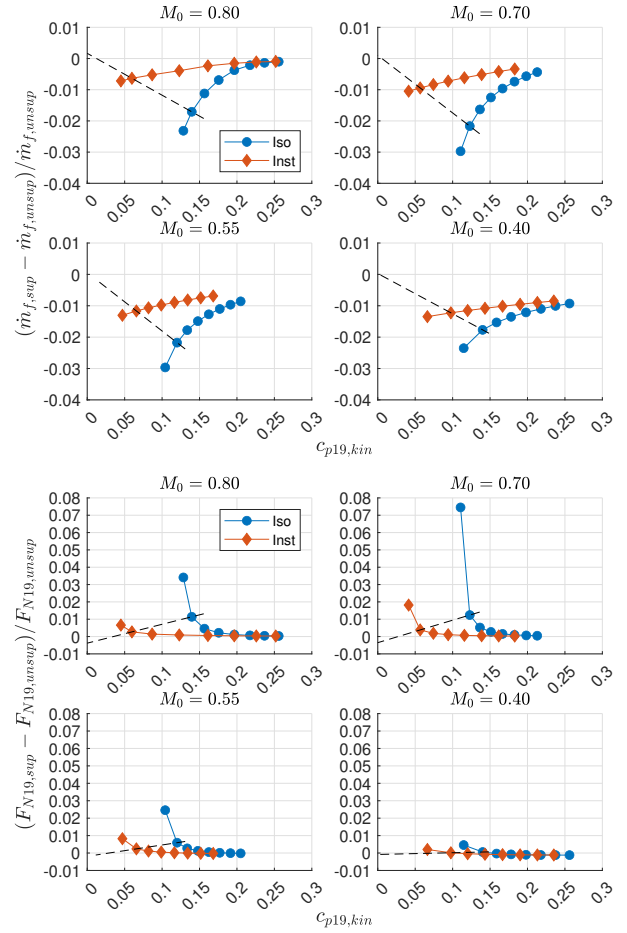


FIG 20. Fan mass flow and net thrust defect using the $c_{p19,kin}$ -data from the thrust variation study as input to the 0-D Gas Dynamics model.

flow, but the impact on thrust production of a TPS, as defined in Equation 26, provided with constant power to the fan, is small in the region of the design point where $FPR \approx 1.3$ and $\alpha = \pm 2^\circ$.

7. SUMMARY

In this work, a parametric geometry model for a turbofan propulsion simulator was set up. A 0-D gas dynamics model was used to calculate the main TPS geometric parameters and the boundary conditions to be applied in the following CFD-simulation campaign. The CFD-based parametric study was carried out using steady-state RANS equations with the Spalart-Allmaras turbulence model and a fully turbulent approach. The TPS was investigated in both the isolated and the installed configuration. NASA's high-speed common research model was employed to represent a large transport airframe with a modern transonic wing design. A mesh convergence study was carried out in order to verify the numerical setup. Two studies were carried out to investigate the influence of various thrust ratings and various angles of attack for range of flight Mach numbers and a fixed Reynolds number. The relevant data was extracted from the simulations

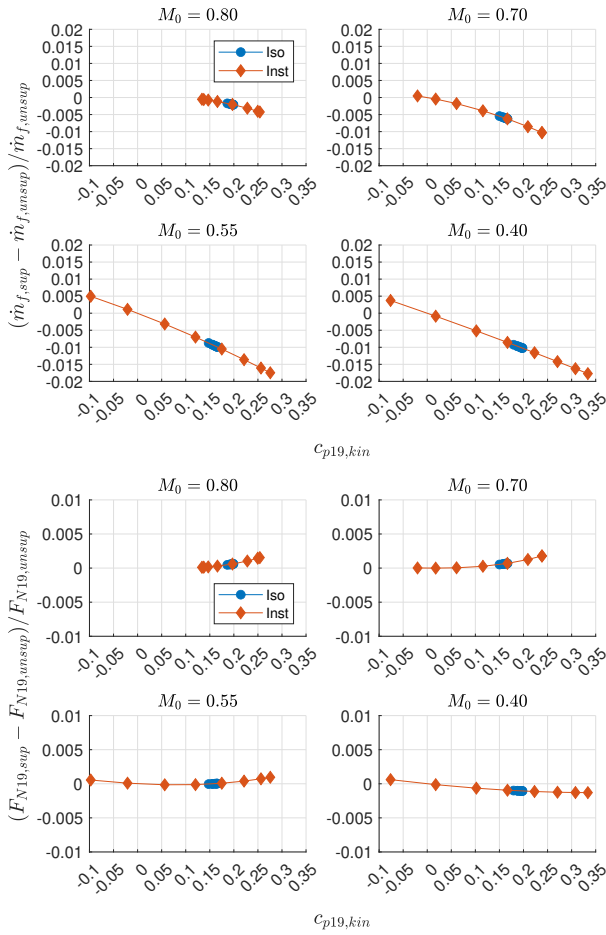


FIG 21. Fan mass flow and net thrust defect using the $c_{p19,kin}$ -data from the VA-study as input to the 0-D Gas Dynamics model.

to describe the operating point and the performance of the TPS. The effects of flow suppression on the isolated and installed configurations were shown and discussed. Finally, the 0-D gas dynamics model was set up to account for flow suppression by means of the fan nozzle exit pressure coefficient $c_{p19,kin}$ to highlight the effects relative to the unsuppressed TPS. The following section answers the research objectives, formulated above.

7.1. Conclusions

The simulation results draw a robust picture of the flow suppression acting on the engine. The fan nozzle exit pressure coefficients increase with the flight Mach number and the thrust rating. For fan pressure ratios approaching unity, $c_{p19,kin}$ does not decrease to zero but rather converges to a positive non-zero value due to the interaction between the nacelle external flow and the fan nozzle efflux. In the thrust variation study, flow suppression was shown to decrease with installation at $\alpha = 0^\circ$. The TPS experienced lower fan nozzle pressure coefficients than the CRM's TFN. Berens et al. [1] reported $c_{p19,kin} = 0.34$ ($c_{p19} = 0.40$) without and $c_{p19,kin} = 0.27$ ($c_{p19} = 0.32$) with a core body at $\alpha = +3^\circ$ and $M_0 = 0.80$ in the installed configura-

tion. For the same conditions and $FPR = 1.3$, the fan nozzle of the TPS experiences pressure coefficients around $c_{p19,kin} = 0.20$. When the fan pressure ratio is decreased to $FPR = 1.0$ ($\alpha = 0^\circ$), the pressure coefficient reduces to $c_{p19,kin} = 0.12$ in the installed configurations. These results highlight the impact of the nacelle and exhaust geometry in comparison to the TFN as well as the role of the operating point of the fan nozzle on the nozzle back pressure and its dependency on the installation effects. Beyond that, $c_{p19,kin}$ shows a strong sensitivity to the angle of attack in the installed configuration. Sufficiently negative angles of attack were shown to produce a negative suppression effect in the installed configuration and even negative pressure coefficients. For angles of attack higher than $+1^\circ$ to $+2^\circ$, $c_{p19,kin}$ rises above the value for the isolated configuration. In the isolated configuration, the effect of the angle of attack on flow suppression is minimal compared to the installed configuration. The use of fan nozzle boundary conditions which are independent of the angle of attack and hence nozzle back pressure creates a discrepancy when comparing the simulation data to the behavior of a TPS in wind tunnel testing. The increase in nozzle back pressure would reduce the fan nozzle mass flow while increasing the total pressure at the fan when the same power is supplied to the fan. The 0-D gas dynamics model predicts a mass flow reduction of up to 3% due to flow suppression, but also an increase in p_{t15} such that FPR increases by up to 0.5%. Hence, the fan operating point shifts due to positive flow suppression. Since mass flow and fan total pressure effects largely compensate each other, the fan net thrust F_{N19} of the TPS does not decrease but increases slightly.

7.2. Recommendations

Future research may focus on a variety of aspects which lay beyond the scope of this work:

- Starting from the gas dynamics model of the TPS, a comparable model for a normal dual-stream turbofan engine should be formulated to resemble more accurately the characteristics of a turbofan engine.
- The engine geometry should be investigated to improve the aerodynamic performance and robustness of the nacelle and the exhaust system.
- A bifurcation shall be incorporated in the isolated and installed configuration to improve the comparability between simulation and reality.
- The numerical approach shall be extended to other turbulence models and improved meshes to improve drag prediction.
- The investigation of the installed configuration in cruise flight shall be performed at angles of attack in the range of $0^\circ < \alpha < +4^\circ$ to approach cruise flight attitudes more closely.

Acknowledgments

This article is based on the Diploma thesis "Effects of Isolated and Installed Turbofan Engine Nozzle Flows on Nozzle Pressure Coefficients for Various Thrust Ratings" by Christoph Wimmer [11], completed at Technische Universität Wien, Research Group for Aircraft Systems.

Contact address:

martin.berens@tuwien.ac.at

References

- [1] Berens, M.; de Rosa Jacinto da Silva, M.; Gerl, B. Simulation of Turbofan Engine Flow Suppression Effects. Deutsche Gesellschaft für Luft- und Raumfahrt - Lilienthal-Oberth e.V., 2022. DOI: [10.25967/570424](https://doi.org/10.25967/570424).
- [2] Christian Sabater Campomanes. External flow effects in the engine/airframe integration testing technique a new thrust/drag bookkeeping approach at the german-dutch wind tunnels. Master thesis, Delft University of Technology, Faculty of Aerospace Engineering, Aerospace Engineering, 2017. <https://resolver.tudelft.nl/uuid:e9e66768-b0be-49f0-970a-1a9adf41a377>.
- [3] SAE International E-33 In Flight Propulsion Measurement Committee. Advanced ducted propulsor in-flight thrust determination. <https://www.sae.org/standards/content/air5450/>.
- [4] E-33 In Flight Propulsion Measurement Committee SAE International. Advanced ducted propulsion in-flight thrust determination, 2014-06. DOI: <https://doi.org/10.4271/AIR5450>.
- [5] W.B. de Wolf. Possibilities and limitations of vhrb and uhbr turbofan simulations in engine/airframe integration wind tunnel experiments. Technical report, National Aerospace Laboratory NLR, 1996. <https://reports.nlr.nl/items/18035f5b-19b2-4d66-bb5d-a6387e2cca08>.
- [6] Tomasz P. Stankowski, David G. MacManus, Christopher T. Sheaf, and Nicholas Grech. Aerodynamic Interference for Aero-Engine Installations. DOI: [10.2514/6.2016-0766](https://doi.org/10.2514/6.2016-0766).
- [7] E-33 In Flight Propulsion Measurement Committee SAE International. In-flight thrust determination, 2017-12. DOI: <https://doi.org/10.4271/AIR1703A>.
- [8] J. Vassberg, M Dehaan, Melissa Rivers, and Richard Wahls. Development of a common research model for applied cfd validation studies. 08 2008. DOI: [10.2514/6.2008-6919](https://doi.org/10.2514/6.2008-6919).
- [9] MIDAP Study Group. Agardograph no.237, guide to in-flight thrust measurement of turbojets and fan engines. Technical report, AGARD, 1979.
- [10] International Organization for Standardization. Standard atmosphere, 1975. ISO 2533:1975. <https://www.iso.org/standard/7472.html>.
- [11] Christoph Carl Wimmer. Effects of Isolated and Installed Turbofan Engine Nozzle Flows on Nozzle Pressure Coefficients for Various Thrust Ratings. Diploma thesis, Technische Universität Wien, 2025. DOI: [10.34726/hss.2025.113860](https://doi.org/10.34726/hss.2025.113860).
- [12] Christoph Carl Wimmer. Parametric nacelle and exhaust geometry model for a turbofan propulsion simulator. Projektarbeit, Technische Universität Wien, 2025.
- [13] Robert Christie, Alexander Heidebrecht, and David MacManus. An automated approach to nacelle parameterization using intuitive class shape transformation curves. Journal of Engineering for Gas Turbines and Power, 139, 11 2016. DOI: [10.1115/1.4035283](https://doi.org/10.1115/1.4035283).
- [14] Parametric Geometry and CFD Process for Turbofan Nacelles, volume Volume 1: Aircraft Engine; Fans and Blowers; Marine of Turbo Expo: Power for Land, Sea, and Air, 06 2016. DOI: [10.1115/GT2016-57784](https://doi.org/10.1115/GT2016-57784).
- [15] Ed Obert and Ronald Slingerland. Aerodynamic design of transport aircraft. IOS Press Delft University of Technology, Faculty of Aerospace Engineering, Section Design of Aircraft and Rotorcraft, Amsterdam [Delft], 2009. ISBN: 1607504073.
- [16] The MathWorks Inc. Matlab version: 23.2.0.2428915 (r2023b), 2023b. <https://www.mathworks.com>.
- [17] The MathWorks Inc. Symbolic math toolbox: Version 23.2 (r2023b), 2023. <https://www.mathworks.com>.
- [18] Siemens Digital Industries Software. Simcenter STAR-CCM+, version 2021.3, Siemens 2021. <https://www.plm.automation.siemens.com/global/en/products/simcenter/STAR-CCM.html>.
- [19] Jai Ahuja, Chung Hyun Lee, Christian Peron, and Dimitri N. Mavris. Comparison of overwing and underwing nacelle aeropropulsion optimization for subsonic transport aircraft. Journal of Aircraft, 61(2):638–653, 2024. DOI: [10.2514/1.C037508](https://doi.org/10.2514/1.C037508).
- [20] P. SPALART and S. ALLMARAS. A one-equation turbulence model for aerodynamic flows. DOI: [10.2514/6.1992-439](https://doi.org/10.2514/6.1992-439).
- [21] Siemens Digital Industries Software. Simcenter STAR-CCM+ User Guide v. 2021.3, Siemens 2021.
- [22] AIAA. Drag prediction workshop IV, 2009. <https://aiaa-dpw.larc.nasa.gov/Workshop4/workshop4.html>.