



Numerical investigation of a Coandă-based fluidic thrust vectoring system for subsonic nozzles

Nils Schwagerus¹ · Marcel Stößel¹ · Michael Krummenauer² · Dragan Kožulović¹ · Reinhard Niehuis¹

Received: 23 February 2023 / Revised: 20 June 2023 / Accepted: 3 August 2023 / Published online: 5 September 2023
 © The Author(s) 2023

Abstract

The numerical investigation of a novel subsonic thrust concept for fluidic thrust vectoring (FTV) of jet engines is the subject of this publication. FTV possibly offers advantages over conventional mechanical thrust vector control such as decreased complexity, mass and maintenance costs. The operating principle of the FTV nozzles under investigation at the Institute of Jet Propulsion is based on the Coandă effect. The applied concept of thrust vectoring uses dedicated secondary flow channels which are mounted in parallel around the nozzle inner cone or the nozzle wall. If required, bleed air, provided by extraction from the engine compressor or from an external source, is injected at a specific nozzle pressure ratio at the nozzle throat A_g through these secondary ducts, resulting in the redirection of the secondary jet towards the convex Coandă surface. The interaction between the primary mass flow and the secondary jet leads to a redirection of the primary exhaust mass flow of the engine and thus to a vectoring of the exhaust flow. In this paper, the influence of different nozzle geometric parameters and different operating points are investigated within an extensive parametric study applied to a convergent two-dimensional thrust vectoring nozzle using computational fluid dynamics tools. Thrust vector deflection of up to 20° at a maximum secondary to primary mass flow ratio of 10% is achieved. Reducing this mass flow rate to 5% still yields vectoring angles of up to 15° whereby similar deflection angles compared to conventional mechanical thrust vectoring systems are achievable.

Keywords Fluidic thrust vectoring · CFD · Gas turbine · Coandă · Thrust vectoring

List of symbols

Greek symbols

α	Angle of attack ($^\circ$)
β	Thrust vector angle ($^\circ$)
ϕ	Roll angle ($^\circ$)
Π_D	Total pressure ratio (–)
ψ	Yaw angle ($^\circ$)
θ	Pitch angle ($^\circ$)
ζ	Total pressure loss coefficient (–)

Latin symbols

A	Area (m^2)
C_{w0}	Zero-lift drag coefficient (–)
D	Diameter (m)

D_{Jet}	Jet exit diameter (m)
F	Thrust (N)
$F_{r,\text{max}}$	Maximum resulting vectoring thrust (N)
h	Height (m)
k	Turbulent kinetic energy (m^2/s^2)
$\dot{m}$	Mass flow (kg/s)
p	Pressure (Pa)
q	Immersion depth (mm)
R	Coanda-radius (mm)
T	Temperature (K)
u	Axial velocity component (m/s)
U_{Jet}	Jet velocity at jet exit (m/s)
v	Radial velocity component (m/s)
x	Coordinate in x -direction (–)
y	Coordinate in y -direction (–)
z	Coordinate in z -direction (–)

Subscripts

3	Compressor exit plane
5	Turbine exit plane
7	Nozzle inlet plane
8	Nozzle throat plane
9	Nozzle exit plane

✉ Nils Schwagerus
 nils.schwagerus@unibw.de

¹ Department of Aerospace Engineering, Institute of Jet Propulsion, University of the Bundeswehr Munich, Werner-Heisenberg-Weg 39, 85577 Neubiberg, Germany
² Bundeswehr Technical Center for Aircraft and Aeronautical Equipment, WTD 61, Flugplatz, 85077 Manching, Germany

Prim	Primary
Sec	Secondary
Total	Total value

Abbreviations

2D	Two-dimensional
3D	Three-dimensional
ADW	AeroDesignWorks
AHSJ	Axisymmetric hot subsonic jet
CFD	Computational fluid dynamics
DoE	Design of experiments
FTV	Fluidic thrust vectoring
IJP	Institute of jet propulsion
LO	Low observability
MTV	Mechanical thrust vectoring
NATO	North Atlantic Treaty Organization
NLRC	NASA Langley Research Center
NPR	Nozzle pressure ratio
PIV	Particle image velocimetry
RANS	Reynolds-averaged Navier–Stokes
RCS	Radar cross section
SFC	Specific fuel consumption
V/STOL	Vertical short-take-off and landing

1 Introduction

Propulsion concepts with integrated thrust vectoring have been subject of investigations in the past and are already implemented in various forms in selected aerospace applications. Examples frequently mentioned in the literature are the Rockwell-MBB X-31 and the Lockheed Martin F-22. The Rockwell-MBB X-31 has a fixed nozzle design and, downstream of the conventional nozzle exit, pivoting flaps for three-dimensional deflection of the exhaust jet. Test flights have already been carried out on this aircraft without its tail fin [1, 2]. A reduction of the fin area or complete substitution by a thrust vectoring nozzle can lead to a considerable reduction of the aerodynamic drag, the aircraft mass as well as the radar cross section (RCS). The Lockheed-Martin F-22, on the other hand, relies on rectangular shaped and mechanically adjustable nozzles for a two-dimensional deflection of the thrust jet. The aforementioned and other thrust vectoring

concepts still in use in aviation today have in common that they rely on mechanical deflection components. According to Alcorn et al. [2], the replacement of conventional aerodynamic control surfaces brings enormous advantages, which include the improvement of low-observability (LO) as well as the reduction of aircraft mass and zero-lift drag C_{w0} . This can result in higher specific thrust and increased range or payload [2]. Thus, this technology can provide access to shorter runways for both civil and military applications. Furthermore, the ability to generate a specific turning moment through the thrust vectoring nozzle in the event of a failure of conventional control surfaces increases the operational safety of the aircraft. For high performance aircraft, there is also the possibility of providing manoeuvrability beyond the stall region (post-stall), which leads to increased possible angles of attack α and an extension of the flight envelope. In Table 1 a sample of past and current developments on mechanical thrust vectoring systems is shown.

The origin of fluidic thrust vectoring technology lies in space flight. For example, the LGM-30 Minutemen, which has been in service in various versions since the 1960s, already used FTV to alter its flight attitude. However, the advantages of mechanical thrust vectoring described in the previous section go hand in hand with disadvantages, which is why research into fluidic thrust vectoring concepts has also been going on in the aerospace sector for some time. Fluidic thrust vectoring has advantages over mechanical thrust vectoring, which are expressed in a lower weight, a reduced complexity and thus also reduced maintenance costs [5]. Since fluidic thrust vectoring nozzles have a fixed external geometry, the RCS can be decreased compared to that of a moving mechanical thrust vectoring nozzle. In addition, fluidic thrust vectoring offers the potential to reduce aerodynamic control surfaces. Aerodynamic trimming with trim tabs, which is necessary for certain phases of flight, can be replaced and thus reduces aerodynamic trim drag. These and other factors have contributed to the motivation for this and previous scientific work undertaken in the area of fluidic thrust vectoring for aircraft applications [6–15].

Fluidic thrust vectoring concepts can be subdivided into shock vector control (SVC), throat shifting (TS), dual throat nozzle (DTN) and Coandă-coflow and -counterflow concepts. Deere et al. [6, 7, 9] and Flamm et al. [10] carried

Table 1 Overview of conventional mechanical thrust vectoring concepts, Loria et al. [1], Anderson [3], Klußmann and Malik [4]

Aircraft	Year	Type	Attitude control	$\beta_{\max}$ (°)	$\dot{\beta}$ (°/s)	$F_{r,\max}$ (kN)
YF-22/F-22	1986/2005	2D	θ, ϕ	20	110	2 * 34,7
F/A-18 HARV	1987	3D	θ, ψ, ϕ	25	60	2 * 21,9
F-15 ACTIVE	1988/1991	3D	θ, ψ, ϕ	20	80	2 * 17,8
Rockwell-MBB X-31	1990	3D	θ, ψ	35	60	
X-62 VISTA	1992	3D	θ, ψ	20	60	1 * 27,8
EJ230	–	3D	θ, ψ, ϕ	23	110	2* 30.6

out comparative numerical and experimental investigations on the abovementioned FTV concepts. The respective advantages and disadvantages as well as the operational constraints of the mentioned concepts are summarised in Table 2. In the principle figures of Table 2, the primary mass flow is represented by black vectors and the secondary mass flow by blue vectors. In addition to the mentioned investigations, research projects in the field of innovative control effectors for aircraft manoeuvring have been funded within the framework of the North Atlantic Treaty Organization STO AVT-239 Task Group (published in November 2020) in cooperation with universities and industrial partners, which further underlines a current interest in fluidic thrust vectoring technology [16].

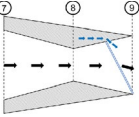
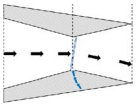
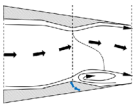
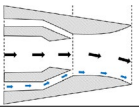
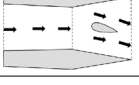
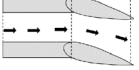
In the work presented within this paper, exhaust jet deflection using the coflow Coandă effect is investigated. Since the nozzle flow is required to remain at subsonic velocities, the concepts SVC, TS and DTN already investigated in the literature cannot be applied. Subsonic co- and counterflow Coandă nozzles have also been investigated in the literature. However, these usually feature a convergent-divergent geometry and are thus inherently not ideal for subsonic use. The divergent section acts as a diffuser in a purely subsonic-operated nozzle, thus leading to a deceleration of the flow. In addition, a too large opening angle of the divergent nozzle part can lead to flows detached from the nozzle wall [25]. Furthermore, the prevailing low pressure in the divergent section for subsonic nozzle flow can lead to

a reversal of the jet direction. This is especially the case at low secondary to primary mass flow ratios, as preliminary studies undertaken in preparation for this paper have shown. This effect has also been mentioned by Bougas and Hornung [26]. Aforementioned phenomena would consequently lead to a reduction in the thrust of the jet engines, which are thus unsuitable for propulsion applications. Therefore, a purely convergent nozzle, in which the Coandă radii are arranged around the inner cone, is chosen as the primary nozzle configuration investigated in the present work. The question to be answered is whether fluidic thrust vectoring in combination with jet engines of aircraft flying at subsonic velocities can be implemented with this method. For this purpose, the influence of geometric as well as fluidic parameters of such nozzles on the deflection of the thrust jet and other characteristic parameters will be investigated in a comprehensive parametric study.

2 Coandă-effect

In the following sections, the theoretical basics of the Coandă effect are explained. Additionally, the fluidic and geometric parameters relevant to this work are discussed. These influencing variables are combined in the subsequent part to form a comprehensive parametric study for characterising the nozzle operating behaviour.

Table 2 Overview: fluidic and mechanical thrust vectoring methods

	Method and flow regime	Principle	Characteristics	References
Fluidic	Shock vector control (SVC) <i>Supersonic</i>		– Moderate-high thrust losses – Risk of wall impingement – Supersonic flow required – Overexpanded operation	[17]
	Throat shifting/skewing (TS) <i>Supersonic</i>		– Low thrust losses – High vector angles – Moderate-high vectoring efficiency	[18]
	Dual throat (DTN) <i>Supersonic</i>		– Low thrust losses – High vector angles – High vectoring efficiency – Linear attitude	[9, 19]
	Coanda co-/counterflow <i>Sub- and Supersonic</i>		– High vector angles – Linear attitude – Low thrust losses	[20–24]
Mechanical	Variable external geometry <i>Sub- and Supersonic</i>		– High vector angles – Mass inertia of moving parts – Complex airframe integration	
	Variable internal geometry <i>Sub- and Supersonic</i>		– Expensive manufacturing and maintenance process – Increase of RCS	Tab. 1

2.1 Fluidic fundamentals

The Coandă effect, named after Henri Coandă (1886 - 1972), describes the ability of a gas jet to follow a convex surface over a comparatively long distance. This phenomenon was first described by O. Reynolds [27] in 1870 for the flow around a sphere and later by Lafay in 1918 [28], Ackeret in 1926 [29] and finally by Coanda [30–32]. Basic investigations on this were undertaken by Fernholz [32] and are used in this work to describe the Coandă effect. The theoretical operating principle of the Coandă effect can be explained by the interaction of a free jet with static air, which is located between said free jet and a wall. The static air is accelerated due to turbulent momentum exchange with the free jet. The wall prevents the supply of new air, which creates a low-pressure zone that draws the free jet against the wall. A curved wall therefore can lead to a deflection of the free jet. For the case of the deflected free jet, a radial mass force directed outwards and a pressure force maintain equilibrium, as explained by Squire and Fernholz [32, 33]. It is utilised in the context of the present work in the so-called coflow variant for the deflection of the primary jet of the nozzle. In the coflow variant, in addition to the conventional, primary exhaust jet, a secondary jet is generated by injecting compressed air in secondary channels parallel to the flow direction of the primary jet (see Fig. 1 and Coandă co-/counterflow in Table 2). This compressed air can be provided, for example, by the compressor in the form of compressor bleed air, by the bypass or by external sources. In the coflow Coandă nozzles discussed in the literature, the secondary channels are usually located in the outer wall area of the nozzle. Due to the hereby necessary arrangement of the convex surfaces for the Coandă effect, this design always results in a convergent-divergent nozzle geometry, such that in the subsonic operating range, as explained in Sect. 1, velocity and thus thrust losses are to be expected. Contrary to this, a purely convergent thrust vectoring nozzle is extended with

secondary channels around the inner cone in the context of this work, thus maintaining the convergent nozzle geometry.

2.2 Important fluidic and geometric parameters

In the open literature, a large number of geometric and fluidic parameters influencing the jet deflection by means of the Coandă effect are mentioned. According to Fernholz [32], the most important geometric parameters include the nozzle height h , the Coandă radius R as well as the immersion depth or vortex step q (see Fig. 1). These characteristic quantities hold the most significant influence on the deflection angle β and the contact behaviour of the secondary jet with the Coandă radius. The ratio of immersion depth to nozzle height q/h and Coandă radius to nozzle height R/h can be defined as decisive factors in the effectiveness of the deflection of the jet around the Coandă radius. It is important to mention, that in this paper, the value of geometric variable q is varied in negative y direction and thus creates a step (refer to Fig. 1). According to the literature, this step is expected to induce vortex structures which are expected to improve the attachment of the secondary jet to the Coandă surface [35]. In the investigations of Fernholz [32] and Neuendorf et al. [36] the Coandă effect was treated as a two-dimensional case. This simplification is valid only if the nozzle possesses a sufficient size or depth which means that the pressure gradient in circumferential direction can be neglected.

Among the fluidic influencing variables, in literature primarily the injection mass flow $\dot{m}_{\text{sec}}$ and the injection velocity c_{sec} or the injection momentum flux $\dot{I}_{\text{sec}} = \dot{m}_{\text{sec}} \cdot c_{\text{sec}}$ are mentioned. In this work, additionally the jet deflection angle β given as a function of the kinetic energy flux $\dot{E}_{\text{kin,sec}} = \dot{m}_{\text{sec}} \cdot c_{\text{sec}}^2$ is considered. The investigation of the parametric influences in the context of this work further deals with the comparison of mass flow ratio, momentum flux ratio and the ratio of the kinetic energy fluxes of secondary to primary mass flow to describe the deflection angle.

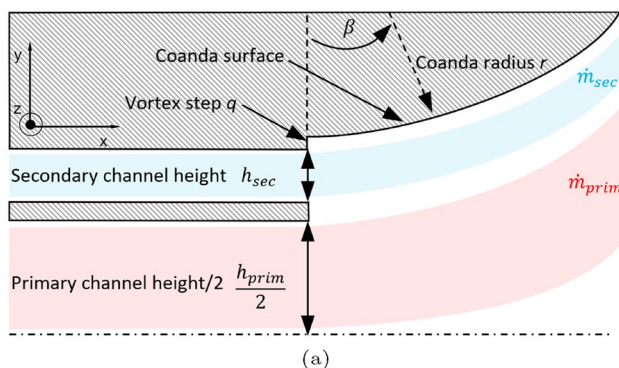


Fig. 1 Coandă effect operating principle (based on [32, 34, 35])

3 Methodology

In this work, an extensive numerical parametric study is carried out with a variation of the characteristic geometric and fluidic parameters (refer to Sect. 2.2). The chosen numerical solver and workflow are described in Sect. 3.3. The parameter study is carried out entirely with a convergent nozzle (Fig. 2), whose properties are described in Sect. 3.1.

3.1 Concept development of the fluidic thrust vectoring nozzles

The fluidic thrust vectoring nozzles discussed in this paper are designed using the fluidic and geometric boundary

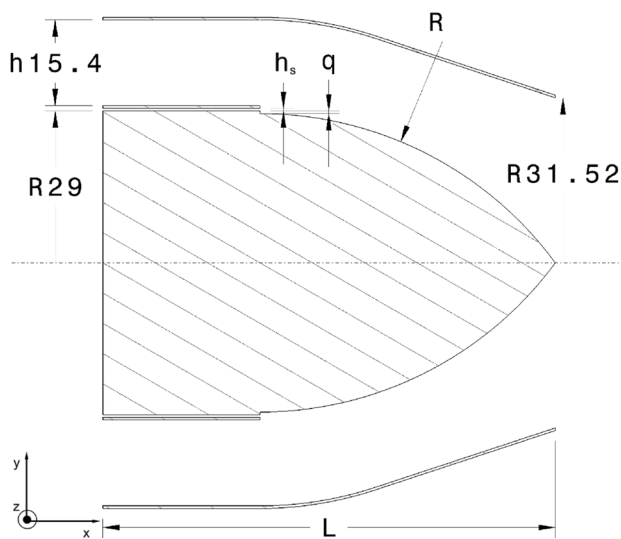


Fig. 2 Sectional view (x - y -plane) of the three-dimensional convergent coflow Coandă nozzle configuration

conditions of the AeroDesignWorks B300F jet engine [37]. The operating points investigated and the associated aerodynamic properties relevant to this work are summarised in Table 3. The turbine outlet flow conditions (plane 5) are used as nozzle inlet boundary conditions (plane 7). The geometric dimensions of the B300F result in the inlet area $A_5 = A_7$, which is adopted unchanged for the three-dimensional fluidic thrust vector nozzle, as is the contraction ratio A_8/A_7 . It should be emphasised that in this work a three-dimensional and a thereof derived two-dimensional model of the fluidic coflow Coandă thrust vector nozzle is investigated. The two-dimensional thrust vectoring nozzle is considered at first, since it allows for a comprehensive parametric study with a significantly lower time expenditure compared to the three-dimensional nozzle geometry. In the two-dimensional case, the nozzle is modelled as an infinitely long nozzle with

a cylindrical Coandă radius in the z -direction. According to Fernholz [32], however, the results of this two-dimensional nozzle are not completely allowed to be compared to the behaviour of the three-dimensional nozzle. Due to the additional pressure gradient acting in the circumferential direction in the three-dimensional case, a lower jet deflection is expected here compared to the two-dimensional case, in which only a transverse pressure gradient prevails. However, the two-dimensional modelling is suitable to investigate the behaviour of quasi two-dimensional thrust vectoring nozzles (no infinite length in z -direction, but delimitation by vertical closed walls) with a rectangular outlet cross-section (2D thrust vectoring). Especially for aircraft with flat fuselages, as it is often the case with unmanned aerial vehicles, this type of fluidic thrust vectoring offers straightforward integrability compared to mechanical thrust vectoring. The question of how far the results of the parametric study of the two-dimensional nozzle can be transferred to the three-dimensional nozzle is a further subject within this work. To model the two-dimensional fluidic thrust vectoring nozzle utilising the boundary conditions given by the B300F, a constant relative mass flow density $\sigma_5 = \frac{\dot{m} \cdot \sqrt{T_5}}{p_{t,5} \cdot A} \cdot \sqrt{\frac{R}{\kappa}}$ is assumed. The convergent nozzle has secondary channels which run circumferentially around the inner cone parallel to the primary flow direction and inject the secondary mass flow $\dot{m}_{\text{sec}}$ at the nozzle throat A_8 . In the three-dimensional case, these annular channels are each divided into four segments by four separate sectors which are offset to each other by 90° . With these four segments, in the three-dimensional case it is possible to generate a fluidic thrust deflection around all three spatial axes (3D thrust vectoring).

3.2 Parametric study

The geometric parameters varied in the parametric study within this paper are shown in Fig. 2 and the range of values of the respective geometric parameters can be found in Table 4. For the definition of the boundary conditions, operating data of the engine, which were collected by Jäger [38] in the course of experimental investigations at the IJP, are used (refer to Table 3). Within the framework of the parametric study carried out in this work, a large number of design and operating points are varied and simulated. The

Table 3 Investigated operating points ADW B300F, [38]

	Operating points				Units
Nr	1	2	3	4	[-]
RPM	97800	80000	60000	35000	(U/min)
F_s	230	116	50	14.4	(N)
$\dot{m}$	0.53	0.40	0.26	0.13	(kg/s)
p_∞	94000	94000	94000	94000	(Pa)
T_∞	288.15	288.15	288.15	288.15	(K)
Ma_5	0.56	0.44	0.30	0.17	(-)
$p_{t,5}$	147261	120549	106,169	97,839	(Pa)
$T_{t,5}$	958.90	813.94	800.02	877.48	(K)
A_5	0.0035	0.0035	0.0035	0.0035	(m ²)
σ_5	0.0318	0.0270	0.0198	0.01125	($\sqrt{K} \cdot s/m$)

Table 4 Parametric space of the investigated geometric nozzle parameters

	Range of values				Units
Coandă radius R	70	100	130		[mm]
Height of secondary channel h	0.5	1	2		[mm]
Immersion depth q	0	1	2	4	[mm]

number of investigated design and operating points for the two-dimensional convergent nozzle amounts to approximately 1000, which allows a complete coverage of the entire geometric (see Table 4) and operating point parameter space. For each of the four operating points shown in Table 3, the secondary mass flow ratio is varied in steps of $\dot{m}_{\text{sec}}/\dot{m}_{\text{prim}} = 0; 0.01; 0.025; 0.05; 0.075; 0.1$ to investigate the influence of the mass flow ratio and the absolute operating point of the engine on the thrust vector deflection characteristics. Additionally, the parametric study includes the variation of the geometric quantities such as the Coandă radius R , the secondary duct height h (for the three-dimensional nozzle this height becomes the secondary duct area A_{sec}) and the Coandă immersion depth q illustrated in Fig. 2.

3.3 Numerical modelling

In the following section, the methodology for the spatial discretisation of the computational domain as well as the settings selected within the numerical flow solver ANSYS FLUENT for the steady-state simulations are described. Subsequently, the mesh independence of the simulation results from the spatial discretisation of the computational grid used in the further course of this paper is discussed based on a grid convergence study.

The flow solver settings used in this work are based on the Reynolds-averaged Navier–Stokes equations (RANS). For the solution of the steady RANS equations applied in this paper and modelling of the Reynolds stress tensor τ_{ij} , the $k - \omega - SST$ turbulence model is used due to its good predictions in adverse pressure gradients and separating flow [39]. Air is modelled as an ideal gas. Since temperatures of $T > 900 \text{ K}$ can occur in the exhaust gas jet, the specific heat capacity c_p is modelled as temperature-dependent. For this purpose, the NASA 9-coefficient polynomial functions are used to describe the temperature dependence [40]. For modelling the viscosity, the description according to Sutherland is used [41].

To enable automatic mesh generation for the large quantity of design- and operating points investigated within this paper, ANSYS Workbench is used. A bidirectionally communicating plug-in enables the ANSYS Workbench to read in and modify geometric parameters of the fluidic thrust vectoring nozzles previously created and fully parameterised in the CAD tool CATIA V5. In this way, the geometric and fluidic parameter spaces can be specified in tabular form, automatically updated in the CAD environment and processed in the Workbench environment. The Workbench workflow includes meshing, solving of the numerical flow problem and the creation and readout of post-processing data for further evaluation in TECPLOT and MATLAB.

ANSYS Meshing is used for meshing the two-dimensional geometries, FLUENT meshing is used for meshing

the three-dimensional geometry. With the same resolution of the boundary layer and an equally fine resolution of areas relevant to the physics of flow (e.g. the shear layer between the secondary jet and the primary exhaust flow, defined by custom bodies of influence), the number of cells of the computational grid in the three-dimensional case exceeds that of the two-dimensional case by a factor of up to ~ 200 . This circumstance demonstrates the necessity of reducing the model complexity by adapting the nozzle to a two-dimensional geometry in order to decrease the time required to solve the numerical simulations, within a comprehensive preliminary study.

To ensure a sufficient resolution of the viscous sublayer of the turbulent boundary layer and thus an accurate calculation of the friction drag, a dimensionless wall distance of $y^+ \approx 1$ is realized [42]. The wall distance y of the first grid cell can be estimated using the Eq. 1 and iteratively adjusted hereafter.

$$y = \frac{y^+ \mu}{\sqrt{\rho \tau_w}}. \quad (1)$$

For the two- and three-dimensional computational grids, a grid convergence study is carried out applying the Grid Convergence Index by Roache et al. [43, 44]. For this purpose, the deflection angle β and the resulting thrust F were evaluated using three different meshes. Hereby, the values of $GCI_{\text{fine}}^{21} = 1.63\%$ and 0.0048% are achieved respectively and suggest that a fine enough mesh is selected. In a previous step, the required wall distance y of the first grid cell was already determined iteratively, such that the boundary layer resolution is not changed further in the grid convergence study. A total cell number of approximately $35 \cdot 10^4$ is obtained with these settings.

4 Validation of the numerical simulations

To validate the numerical setup chosen in this paper, experimental comparison data from the NASA Langley Research Center (NLRC) Turbulence Modeling Resource database [45–47] are used. The NLRC hereby provides a public database with both experimental and additionally simulated comparative data for the verification and validation of the flow solvers used in respective applications. In this publication, the NLRC comparison case of an axisymmetric hot subsonic jet (AHSJ) is used. This validation case is suitable due to the high prevailing temperature in the thrust vector nozzle ($T_{t5, \text{max}, B300F} \approx 958 \text{ K}$) and the correct depiction of the shear layer behaviour between primary and secondary air mass flow, which is an essential aspect for thrust jet redirection.

The hot jet in the experiment of this validation case documented by Bridges and Wernet [48, 49] exhibits a nozzle exit Mach number of $Ma_{jet} = 0.376$ and is surrounded by stationary ambient air (see Fig. 3a). To increase the compatibility of the associated simulated comparison case with different CFD codes, a Mach number of $Ma_{ref} = 0.01$ was imposed by the NLRC as a boundary condition for the ambient flow in the numerical simulation, since some CFD codes exhibit numerical instabilities at farfield boundary conditions with fully quiescent flow. The influence of this low Mach number on the hot jet flow quantities turns out to be negligible [50]. Due to the axisymmetric geometry

of the test case, a computational grid rotated by one degree with periodic boundary conditions can be used within the numerical simulation to accelerate the computation time while improving the spatial discretisation compared to a three-dimensional simulation. Experimentally, particle image velocimetry (PIV) was used by the authors mentioned above to determine velocity and turbulence profiles at selected locations parallel and perpendicular to the flow direction. In addition, numerical simulations with the NASA flow solvers CFL3D [51] and WIND [52] were carried out by the NLRC, providing both experimental and numerical comparative data.

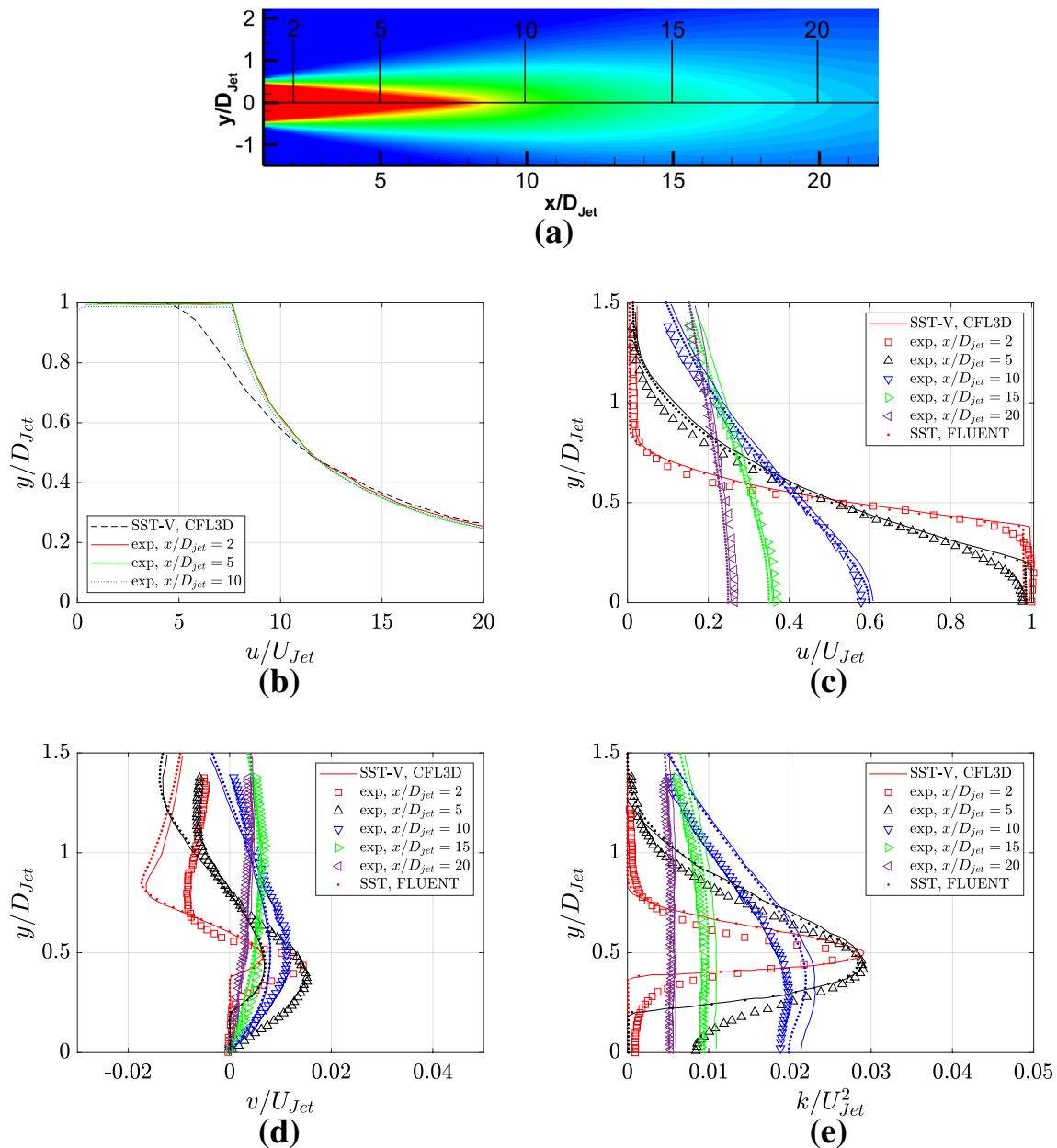


Fig. 3 Comparison of experimental and simulated data [48] with solver setup used in this work

Figure 3 illustrates the NLRC experimental and simulated data together with the results generated in this work using the FLUENT flow solver. In the course of validating the settings of the ANSYS FLUENT flow solver, it was found that the $k - \omega - SST$ turbulence model generally exhibits good agreement with the simulations performed by the NLRC. However, it should be mentioned that in some cases small deviations from the experimentally determined flow variables prevail, as can be seen in Fig. 3a. The axial velocity component u normalised over the nozzle exit velocity U_{Jet} is used by all simulations from $x/D_{\text{Jet}} \approx 5$ to $x/D_{\text{Jet}} \approx 7.5$ is overestimated by a maximum of 25%. With increasing distance from the nozzle exit plane, this deviation decreases to less than 3%. However, it should be emphasised here that the simulations carried out in this work and those of NASA agree very well. In addition, Bridges and Wernet [48, 49] explicitly point out an uncertainty in the measurement results that depends on the axial position and decreases in the direction of flow. Further consideration of the velocity profiles in both radial and axial directions in Fig. 3b shows very good agreement between the experimental and all simulated data. Also when considering the turbulent kinetic energy k in Fig. 3d, the simulated and experimental data agree well qualitatively and quantitatively.

5 Evaluation and analysis of the simulation results

In this section, the results of the detailed parametric study of the two-dimensional convergent fluidic coflow Coandă thrust vector nozzle are discussed initially. Here, the influences of the fluidic and geometric parameters are examined, for example, on quantities such as the deflection angle β or quantities describing the efficiency (total pressure ratio Π_D and total pressure loss coefficient ζ).

Within the numerical parametric study of the convergent fluidic thrust vectoring nozzle, about 1000 different parameter combinations are calculated. In the following sections, it is explained which of the geometric and fluidic parameter combinations have the most important influence on the thrust jet deflection of the convergent thrust nozzle.

5.1 Characterisation of different deflection zones

In Fig. 4 the course of the deflection angle β is depicted over the mass flow ratio $\dot{m}_{\text{sec}}/\dot{m}_{\text{prim}}$ at a primary mass flow of $\dot{m}_{\text{prim}} = 0.53 \text{ kg/s}$ and without a Coandă immersion depth ($q = 0 \text{ mm}$). Furthermore, it is distinguished between three cases with different Coandă radius (R70, R100 and R130) and a constant secondary channel height (h0.5). In the aforementioned figure, three zones with respectively different gradients of the deflection angle curve can be identified. In

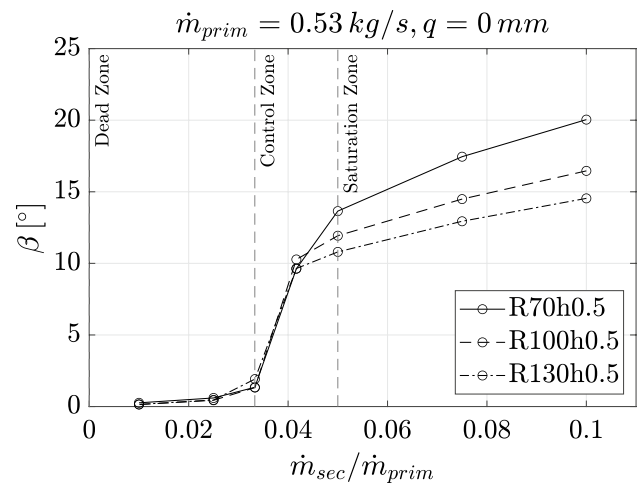


Fig. 4 Trend line of the thrust vector angle (dead zone, control zone, begin of saturation zone)

the first zone, often referred to in the literature as the *Dead Zone*, the secondary mass flow is too low to generate a jet adhering to the Coandă radius. In this area, the Coandă effect is thus not functional. In the following so-called *Control Zone*, a strong increase of the deflection angle as a function of the mass flow ratio can be seen. With further increasing mass flow ratios, the change of the deflection angle decreases until a hypothetical *Saturation Zone* is reached. In this zone, an increase in the secondary mass flow leads only to small changes in the deflection angle. These observations regarding the deflection behaviour as a function of mass flow fraction in the coflow Coandă method were also observed by Mason [20], among others.

For the practical application of the coflow Coandă effect, this means that the *Dead Zone* must be exceeded very quickly if the desired thrust vectoring is supposed to avoid a hysteresis effect. This is the only way to minimise the time difference between the deflection command and the actual deflection of the thrust vector with the desired angle and thrust.

5.2 Influence of the mass flow ratio, R and h on the deflection angle

In Fig. 5a the achieved deflection angle β is plotted against the mass flow ratio of secondary to primary mass flow $\dot{m}_{\text{sec}}/\dot{m}_{\text{prim}}$ at a constant primary mass flow of $\dot{m}_{\text{prim}} = 0.53 \text{ kg/s}$ and without a Coandă immersion depth ($q = 0 \text{ mm}$). The mass flow ratio is thereby calculated in five equidistant steps from $\dot{m}_{\text{sec}}/\dot{m}_{\text{prim}} \cdot 100 = 1\%$ to $\dot{m}_{\text{sec}}/\dot{m}_{\text{prim}} \cdot 100 = 10\%$ and is increased incrementally. The identical process is illustrated in Fig. 5b–d with successively decreasing primary mass flows $\dot{m}_{\text{prim}}$. These four figures represent different operating points of the engine (see Table 3).

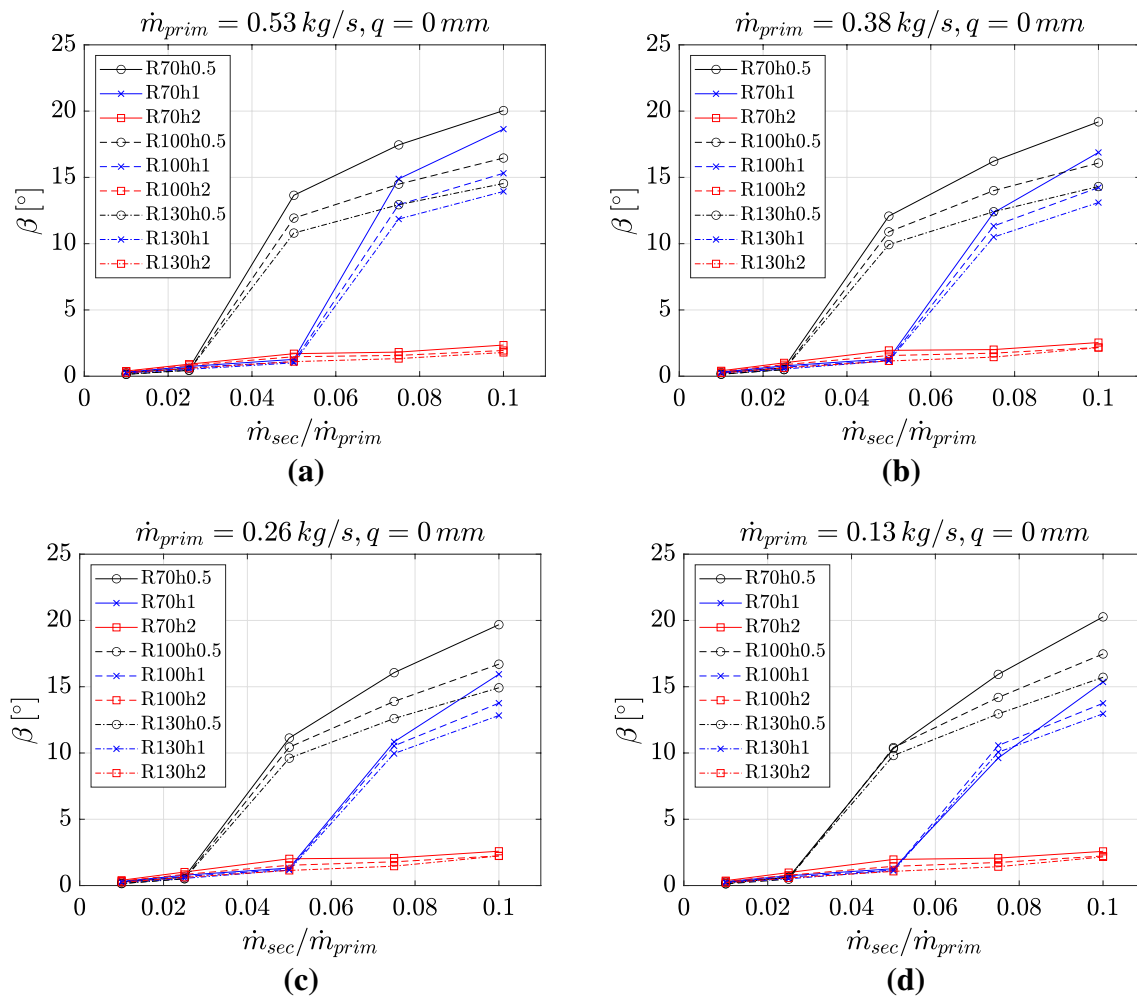


Fig. 5 Plot of the jet deflection angle β versus the mass flow ratio $\dot{m}_{sec}/\dot{m}_{prim}$ at different primary mass flows $\dot{m}_{prim}$ for the convergent nozzle

Based on the four graphs, a trend can be observed that in the parametric space considered, a reduction of the Coandă radius leads to an increase in the deflection angle depending on the mass flow rate considered. For the geometries with a secondary channel height of $h_s = 0.5 \text{ mm}$ (h0.5), a deflection of the primary jet is recognisable from a mass flow ratio of 0.025 independent of the operating point. For the following analysis, the first operating point (see Fig. 5a) is described as an exemplary case.

At a mass flow ratio of 0.05 the deflection angle is $\beta = 13.65^\circ$ for a Coanda radius of $R = 70 \text{ mm}$. For $R = 100 \text{ mm}$ this value decreases to $\beta = 11.93^\circ$ and at $R = 130 \text{ mm}$ further declines towards $\beta = 10.80^\circ$. At $R = 100 \text{ mm}$, β is thus $\Delta_{R100-R70} \approx 12.60\%$ lower than at $R = 70 \text{ mm}$. Regarding $R = 130 \text{ mm}$, β is reduced by $\Delta_{R130-R70} \approx 20.87\%$ compared to $R = 70 \text{ mm}$. With an increasing mass flow ratio of $\dot{m}_{sec}/\dot{m}_{prim} = 0.075\%$, this difference increases at $\beta_{70 \text{ mm}} = 17.45^\circ$, $\beta_{100 \text{ mm}} = 14.49^\circ$ and $\beta_{130 \text{ mm}} = 12.94^\circ$, respectively, to $\Delta_{R100-R70} \approx 16.9\%$

and $\Delta_{R130-R70} \approx 25.85\%$. The maximum thrust deflection angle $\beta = 20^\circ$ is reached for a mass flow ratio of 0.1 and $R = 70 \text{ mm}$. $R = 100 \text{ mm}$ and $R = 130 \text{ mm}$ yield in a thrust deflection angle of $\beta = 16.21^\circ$ and $\beta = 14.83^\circ$, respectively.

A similar behaviour can also be observed for the nozzle geometries with $h_s = 1 \text{ mm}$ (h1). However, it is noticeable that with the increased secondary duct height (with respectively identical primary and secondary mass flows) a relevant deflection of the thrust jet can only be achieved with mass flow ratios $> 5\%$. The increased secondary duct height leads to a reduction of the secondary nozzle exit velocity $c_{8,sec}$ and thus to a reduction of the momentum flow ratio $\dot{I}_{sec}/\dot{I}_{prim}$. As already described in Sect. 2.2, the momentum flow ratio is often mentioned in literature as an important influencing variable on the deflection angle of a coflow Coandă thrust vector nozzle [53]. A further increase of the secondary channel height to $h_s = 2 \text{ mm}$ (h2) leads to hardly any jet deflection angles due to the low secondary nozzle exit velocity. At a mass flow rate of 10%, the thrust jet is only

deflected to a maximum of $\beta_{70\text{mm}} = 2.35^\circ$. Only in case of $h_s = 2\text{ mm}$, barely any difference in the amount of the deflection angle can be observed when varying the Coandă radius.

Regarding the geometries R70h0.5, R100h0.5 and R130h0.5 across the graphs 5a to 5d, it can be seen that for a constant mass flow ratio, the difference between the deflection angles of these geometries is reduced with decreasing primary mass flow. For a primary mass flow of $\dot{m}_{\text{prim}} = 0.13\text{ kg/s}$ this leads to the observation that the deflection angles with the secondary channel height $h_s = 0.5\text{ mm}$ at $\dot{m}_{\text{sec}}/\dot{m}_{\text{prim}} = 0.05$ are almost identical, meaning the Coandă radius has no influence on the deflection angle in these specific cases.

It can be stated that a reduction in the secondary channel height has a large positive effect on the deflection angle of the coflow Coandă nozzle while maintaining an identical secondary mass flow. As the primary mass flow decreases, this effect is amplified. However, decreasing the secondary channel height leads to very high supercritical nozzle pressure ratios of $\text{NPR} = p_{t,7,\text{sec}}/p_{8,\text{sec}} \approx 3.9$ in the cases of $\dot{m}_{\text{prim}} = 0.53\text{ kg/s}$ and $\dot{m}_{\text{sec}}/\dot{m}_{\text{prim}} = 0.1$ and thus to a post-expansion following the secondary channel. This results in maximum secondary jet Mach numbers of $\text{Ma}_{\text{sec}} \approx 2.3$ and thus to high-velocity gradients between the primary and the secondary airflow. Therefore, in the following sections, in addition to the deflection angle, variables for evaluating the nozzle efficiency (total pressure ratio Π_D and total pressure loss coefficient ζ) are examined as significant total pressure losses are expected as a result of the overexpanded secondary nozzle condition. In addition, it can be stated that similar deflection angles can be achieved independent of the primary mass flow and therefore the jet engine operating point, which has positive effects on the application of the investigated fluidic thrust vectoring nozzle as a control element.

5.3 Influence of the height of the Coandă immersion depth q on the deflection angle

As a further geometric parameter, the Coandă immersion depth q is varied from $q = 0\text{ mm} - 4\text{ mm}$, as shown in Fig. 6. For this purpose, the identical design and operating points are considered under variation of q as previously in Sect. 5.2. In literature, a potentially positive effect due to the generation of small-scale vortices induced by the step on the adhesion behaviour of the secondary jet and thus the achievable deflection angle is mentioned [35]. However, this effect could not be confirmed in the scope of this work. On the contrary, an increase in the immersion depth is observed to have a negative influence on the deflection angle, as can be seen in Fig. 6. The deflection angle decreases from a maximum value of $\beta_{q=0\text{mm}} = 20^\circ$ to a value of $\beta_{q=4\text{mm}} = 12^\circ$ at a mass flow ratio of 10%. At a mass flow ratio of 5%, the maximum deflection angle is further reduced from $\beta_{q=0\text{mm}} = 15^\circ$

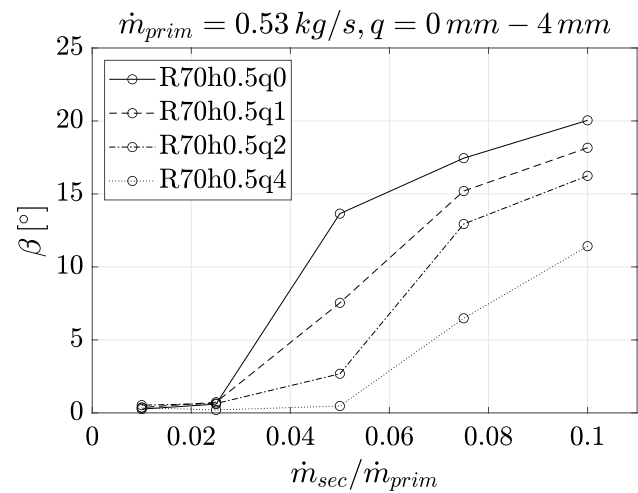


Fig. 6 Vector angle depending on step height

to a value of $\beta_{q=4\text{mm}} = 0.8^\circ$. The q -dependent trend shown here in Fig. 6 occurs in a similar manner with a variation of $\dot{m}_{\text{prim}}$ and h , which is why corresponding illustrations are not presented here.

5.4 Influence on the total pressure ratio

In addition to the behaviour of the deflection angle presented in Sect. 5.2, the total pressure drop behaviour can be examined to quantify the efficiency of a thrust vectoring nozzle. In Fig. 7 the total pressure ratio $\Pi_D = p_{t,9}/p_{t,7}$ is plotted against the deflection angle β . Analogous to Fig. 5, the primary mass flow is reduced in each case. It is noticeable that the total pressure loss is strongly dependent on the primary mass flow. At high primary mass flows, high secondary mass flows prevail in absolute terms. As explained in Sect. 5.2, this leads to high secondary nozzle pressure ratios and thus to supercritical flow through secondary channels of high total inlet pressure $p_{t,7}$. As a result of the expansion and the following supersonic flow of the secondary jet downstream of the secondary channel outlet, high total pressure losses occur in these cases. A comparison of Π_D further shows that an increasing Coandă radius also has a negative influence on the total pressure ratio. This is due to the fact that a larger Coanda radius implies a longer inner cone which again leads to an increased friction drag. However, this effect tapers off with decreasing the primary mass flow.

Furthermore, it should be mentioned that the total pressure drop decreases significantly when the secondary channel height is increased from $h = 0.5\text{ mm}$ to $h = 1\text{ mm}$. The previously described post-expansion occurs here as well, but the increased secondary channel height leads to a reduced secondary channel exit Mach number of $\text{Ma} = 1.52$ and further to a nozzle pressure ratio of $\text{NPR} \approx 2.56$ which poses

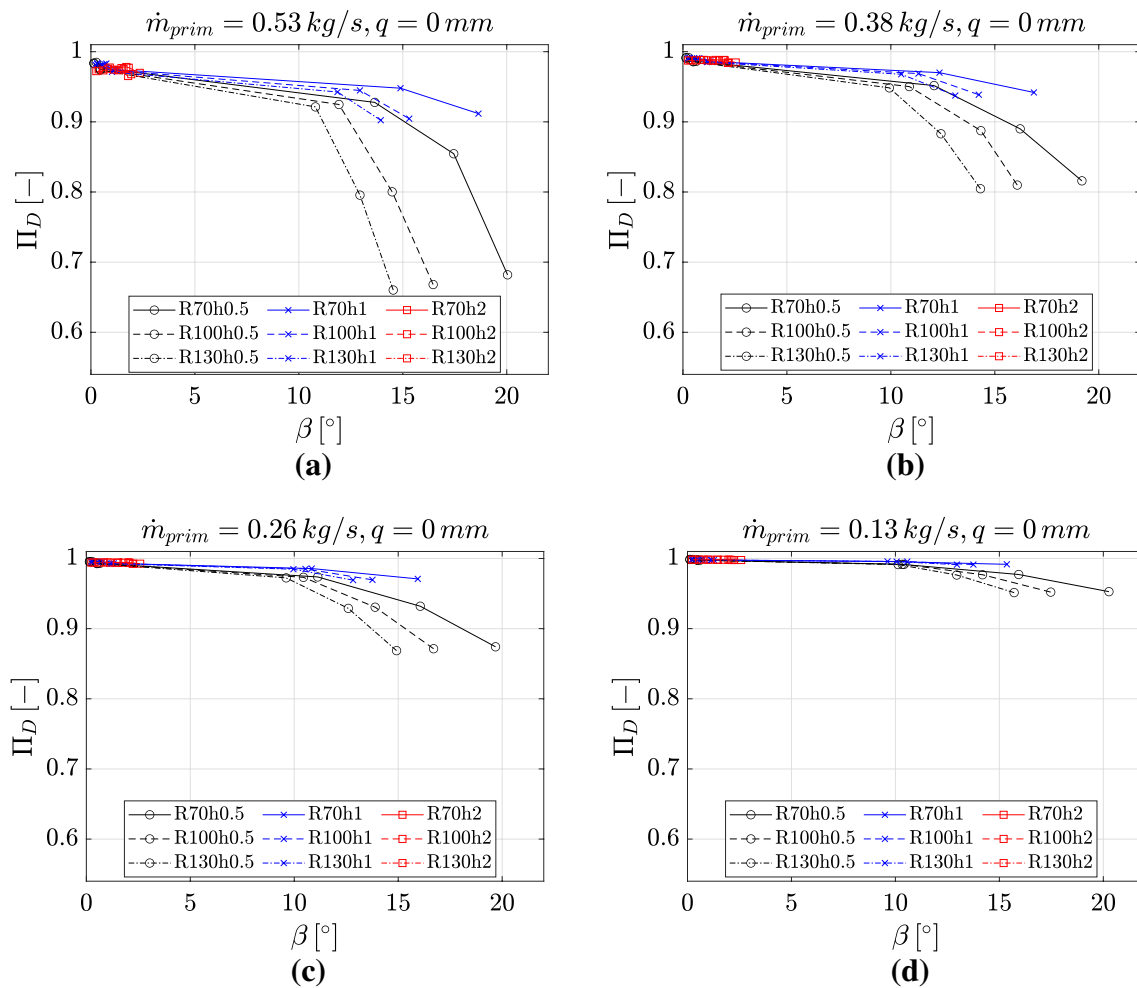


Fig. 7 Plot of the total pressure ratio Π_D versus the deflection angle β at different primary mass flows $\dot{m}_{prim}$ for the convergent nozzle

a significant reduction compared to the NPR observed in Sect. 5.1.

5.5 Comparison of different influence parameters on the jet deflection angle

In the previous sections, an influence of the mass flow ratio on the deflection angle could be determined. However, when considering nozzles with different geometric parameters, it is not possible to make a general statement about the deflection angle to be achieved solely on the basis of a given mass flow ratio. For this purpose, other fluidic variables must be examined in addition to the mass flow ratio. As has been shown, the change in the secondary nozzle exit velocity has a strong influence on the deflection angle to be achieved. This is why the influence of the injection momentum flow $\dot{I} = \dot{m} \cdot c_8$ is investigated in this section. For this purpose, at the position of the throat-section A_8 , the injection momentum flow of the secondary channel is related to that of the primary channel. In Fig. 8 it can be seen that the curves of

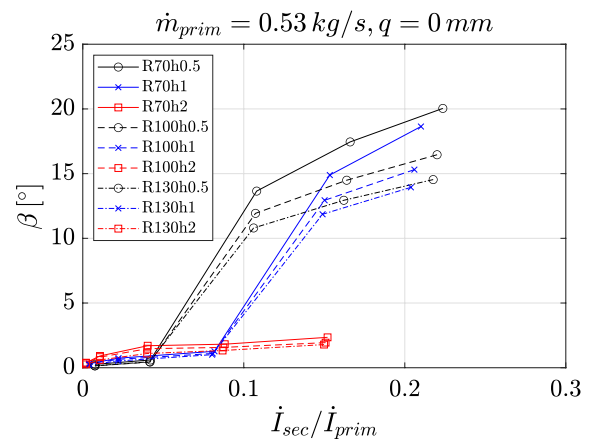


Fig. 8 Influence of the momentum flux ratio on the deflection angle

β are closer together compared to the respective dependence on the mass flow ratio in Fig. 5a. Due to the additional influence of the velocity on the deflection behaviour, the

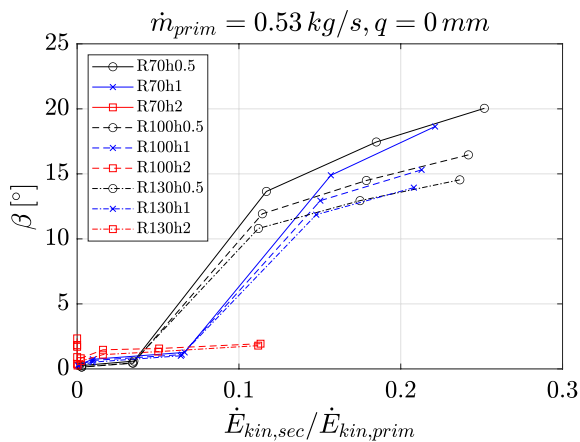


Fig. 9 Influence of the kinetic energy flux ratio on the deflection angle

momentum flow is extended in the next step to the kinetic energy flow $\dot{E}_{kin} = \dot{m} \cdot c_g^2$ (see Fig. 9). The curves of the different geometric and fluidic parameters are now lying even closer together. This reveals a more direct relationship between the ratio of the kinetic energy flows from the secondary to the primary channel, compared to the literature known relations of mass flow and momentum. Especially from a kinetic energy flow ratio of $\dot{E}_{kin,sec}/\dot{E}_{kin,prim} = 0.15$ upwards, a direct correlation to the deflection angle can be observed. It must be taken into account here that the same Coandă radius must be considered in each case. Moreover, this observation is only valid for the area of the *Saturation Zone*. Nevertheless, an improved method of predicting the jet deflection angle more independent of the geometric nozzle parameters can be proposed.

6 Summary and outlook

Within the scope of this work, important findings could be obtained on the influences of geometric and fluidic parameters on the operating behaviour of a fluidic thrust vectoring nozzle that utilises the coflow Coandă effect. Maximum deflection angles of $\beta = 20^\circ$ can be achieved with a two-dimensional convergent fluidic thrust vector nozzle with a mass flow ratio of $\dot{m}_{sec}/\dot{m}_{prim} = 10\%$. Halving the mass flow ratio to 5% results in a reduced deflection angle of $\beta = 15^\circ$. Thus, deflection angles comparable to current mechanical systems used in practice, such as in the Lockheed-Martin F-22, can be achieved (see Table 1). The value of the deflection angle is mostly independent of the operating point of the engine. This circumstance has a positive effect on the implementation of this fluidic thrust vectoring method in practical applications.

The subdivision of the thrust nozzle behaviour into the so-called *Dead Zone*, *Control Zone* and the *Saturation Zone*, which is known from literature, could also be shown by numerical investigations underlying this paper. In addition, the ratio of the kinetic energy flows was identified for a more direct description of the deflection behaviour of a fluidic thrust vector nozzle. In the *Saturation Zone*, this ratio now allows to determine the expected deflection angle at a constant Coandă radius. In literature it is occasionally proposed that an immersion depth could lead to an earlier adhesion of the secondary jet with the Coandă radius and thus to a larger deflection of the thrust jet. However, no positive effect on the deflection behaviour could be observed here. The total pressure losses caused by the immersion depth investigated lead to a loss of energy in the secondary jet and thus to an earlier detachment of the jet from the Coandă radius.

Furthermore, it was found that an increase in the secondary channel exit velocity $c_{8,sec}$ also leads to an increase in the deflection angle. However, due to the thereby proportionally rising high nozzle pressure ratios of the secondary channel, this results in comparatively high total pressure losses.

Based on the previous results, for a future practical application of the fluidic coflow Coandă thrust vector nozzle, a trade-off should be made between the required deflection angle and the associated total pressure loss. Additionally, the focus should be laid on the operating behaviour of the fluidic thrust vectoring nozzle. Here, the effect of the hysteresis effect is of particular importance and should be investigated with unsteady numerical simulation methods. In future investigations, it would also make sense to compare and quantify the difference between the two-dimensional geometry of the fluidic thrust vector nozzle and that of a three-dimensional thrust vector nozzle. In preliminary studies in preparation for this paper, first investigations regarding the performance of a three-dimensional convergent coflow Coandă nozzle have been undertaken. Consistent with the literature, these preliminary investigations have shown that the deflection angle in the three-dimensional case is significantly lower than in the two-dimensional case for the same design and operating point. Thus, the question to what extent three-dimensional fluidic thrust vectoring can be realised with the coflow Coandă effect should be further investigated within a more detailed study. Furthermore, based on the investigations performed in this work, experimental studies using the presented two-dimensional fluidic thrust vector nozzle would be useful for validation purposes and further improvements.

The possibility of ejector-assisted augmentation of the secondary mass flow, for example from ambient air or from a bypass mass flow, could lead to an improvement in efficiency of the thrust vector system investigated in this work, as in this way the required compressor bleed air mass flow could

be reduced. Finally, the influence of the bleed air extraction on the jet engine performance cycle must be carried out.

Funding Open Access funding enabled and organized by Projekt DEAL. Open Access is funded by the DEAL program which is a cooperation between the University of the Bundeswehr Munich and Springer Nature.

Availability of data and materials Not applicable.

Declarations

Conflict of interest The authors have no competing interests to declare that are relevant to the content of this article.

Open Access This article is licensed under a Creative Commons Attribution 4.0 International License, which permits use, sharing, adaptation, distribution and reproduction in any medium or format, as long as you give appropriate credit to the original author(s) and the source, provide a link to the Creative Commons licence, and indicate if changes were made. The images or other third party material in this article are included in the article's Creative Commons licence, unless indicated otherwise in a credit line to the material. If material is not included in the article's Creative Commons licence and your intended use is not permitted by statutory regulation or exceeds the permitted use, you will need to obtain permission directly from the copyright holder. To view a copy of this licence, visit <http://creativecommons.org/licenses/by/4.0/>.

References

- Loria, C.J., Kelly, M., Hatney, R.: X-31A quasi-tailless evaluation. In: IEEE Aerospace Applications Conference. Proceedings (1996). <https://doi.org/10.1109/AERO.1996.499666>
- Alcorn, C.W., Croom, M.A., Francis, M.S., Ross, H.: The X-31 aircraft: advances in aircraft agility and performance. *Prog. Aerosp. Sci.* **32**(4), 377–413 (1995). [https://doi.org/10.1016/0376-0421\(95\)00006-2](https://doi.org/10.1016/0376-0421(95)00006-2)
- Anderson, S.B.: Historical overview of V/STOL aircraft technology, legacy CDMS (1981)
- Klußmann, N., Malik, A.: *Lexikon der Luftfahrt*, 4. auflage edn. Springer, Berlin (2018). <https://doi.org/10.1007/978-3-540-49096-8>
- Hunter, C.A., Deere, K.A.: Computational investigation of fluidic counterflow thrust vectoring. Langley Research Center (1999). <https://doi.org/10.2514/6.1999-2669>
- Deere, K.A.: Computational investigation of the aerodynamic effects on fluidic thrust vectoring. Langley Research Center (2000). <https://doi.org/10.2514/6.2000-3598>
- Deere, K.A.: Summary of fluidic thrust vectoring research conducted at NASA Langley Research Center (2003). <https://doi.org/10.2514/6.2003-3800>
- Ferlauto, M., Marsilio, R.: Numerical simulation of fluidic thrust-vectoring (2016). <https://doi.org/10.13140/RG.2.1.1710.1287>
- Deere, K., Berrier, B., Flamm, J., Johnson, S.: A computational study of a dual throat fluidic thrust vectoring nozzle concept (2005). <https://doi.org/10.2514/6.2005-3502>
- Flamm, J., Deere, K., Mason, M., Berrier, B., Johnson, S.: Design enhancements of the two-dimensional. Dual throat fluidic thrust vectoring nozzle concept (2006). <https://doi.org/10.2514/6.2006-3701>
- Sobester, A., Keane, A.: Multi-objective optimal design of a fluidic thrust vectoring nozzle (2006). <https://doi.org/10.2514/6.2006-6916>
- Friehmelt, H.: Thrust vectoring and tailless aircraft design—review and outlook (1996). <https://doi.org/10.2514/6.1996-3412>
- Clarke, A.J.: The conceptual design of novel future UAV's incorporating advanced technology research components. Doctor of Philosophy (PhD) Thesis, Cranfield University, Cranfield (2011)
- Woods, P.: FLAVIIR—an integrated programme of research for UAV's (2006). <https://doi.org/10.2514/6.2006-3504>
- Gill, K., Wilde, P., Gueroult, R., Crowther, W.: Development of an integrated propulsion and pneumatic power supply system for flapless UAVs (2007). <https://doi.org/10.2514/6.2007-7726>
- North Atlantic Treaty Organization: Innovative control effectors for manoeuvring of air vehicles: summary report of the NATO STO AVT-239 Task Group
- Abeyounis, W.K., Bennett, B.D.: Static internal performance of an overexpanded, fixed-geometry, nonaxisymmetric nozzle with fluidic pitch-thrust-vectoring capability: NASA TP-3645 (1997)
- Miller, D., Yagle, P., Hamstra, J.: Fluidic throat skewing for thrust vectoring in fixed-geometry nozzles (1999). <https://doi.org/10.2514/6.1999-365>
- Ferlauto, M., Marsilio, R.: Numerical investigation of the dynamic characteristics of a dual-throat-nozzle for fluidic thrust-vectoring. *AIAA J.* **55**(1), 86–98 (2017). <https://doi.org/10.2514/1.J055044>
- Mason, M.S., Crowther, W.J.: Fluidic thrust vectoring of low observable aircraft (2004). <https://doi.org/10.2514/6.2004-2210>
- Strykowski, P.J., Krothapalli, A.: The countercurrent mixing layer—strategies for shear-layer control (1993). <https://doi.org/10.2514/6.1993-3260>
- Flamm, J.: Experimental study of a nozzle using fluidic counterflow for thrust vectoring (1998). <https://doi.org/10.2514/6.1998-3255>
- Alvi, F.S., Strykowski, P.J.: Forward flight effects on counterflow thrust vector control of a supersonic jet. *AIAA J.* **37**(2), 279–281 (1999). <https://doi.org/10.2514/2.705>
- Strykowski, P.J., Krothapalli, A., Forliti, D.J.: Counterflow thrust vectoring of supersonic jets. *AIAA J.* **34**(11), 2306–2314 (1996). <https://doi.org/10.2514/3.13395>
- Bräunling, W.J.G.: *Flugzeugtriebwerke*. Springer, Berlin (2015). <https://doi.org/10.1007/978-3-642-34539-5>
- Bougas, L., Hornung, M.: Propulsion system integration and thrust vectoring aspects for scaled jet UAVs. *CEAS Aeron. J.* **4**(3), 327–343 (2013). <https://doi.org/10.1007/s13272-013-0076-x>
- Reynolds, O.: Suspension of a ball by a jet of fluid. *Proc. Manch. Lit. Philos. Soc.* 114–133 (1870)
- Lafay, M.A.: Contribution a l'etude de l'effect Chilowski. *Memoirial de l'Artilerie*, VII, 3 faisceau, 385–392 (1929)
- Ackeret, J.: Grenzschichtabsaugung. *VDI Zeitschrift*, 1153–1158 (1926)
- Coanda, H.: Procédé de propulsion dans un fluide. *Brevet d Invention Gr.6 Cl.2 No.752688* (1932)
- Coanda, H.: Procédé et dispositif pour faire dévier une veine de fluide pénétrant dans un autre fluide. *Gr.5 Cl.3 No.792754* (1934)
- Fernholz, H.-H.: *Zur Umlenkung Von Freistrahlen an Konvex Gekrümmten Wänden (Coanda-Effekt)*: Berlin, Techn. Univ., Habil.-Schr., 1965 (1965)
- Squire, H.B.: Jet flow and its effects on aircraft. *MCB UP Ltd* (1950). <https://doi.org/10.1108/eb031870>
- Kern, F., Bindl, S., Brehm, S., Niehuis, R.: Entwicklung und Konstruktion einer Vorrichtung zur aktiven Verdichterstabilisierung mittels Lufteinblasung am Turbostrahltriebwerk Larzac 04. *Deutscher Luft- und Raumfahrtkongress* (2015)
- Saghafi, F., Banazadeh, A.: Coanda surface geometry optimization for multi-directional co-flow fluidic thrust vectoring, pp. 183–189 (2009). <https://doi.org/10.1115/GT2009-59715>

36. Neuendorf, R., Wygnanski, I.: On a turbulent wall jet flowing over a circular cylinder (1999). <https://doi.org/10.1017/S0022112098003668>
37. AeroDesignWorks GmbH: AeroDesignWorks operating manual B300F. https://www.aerodesignworks.com/wp-content/uploads/OperatingManual_B300F.pdf. Accessed 25 May 2023
38. Jäger, P.: Messbericht Triebwerksversuch Kleingastriebwerk B300F: Interner Institutsbericht Institut für Strahlantriebe (2021)
39. Menter, F.: Zonal two equation k-omega turbulence models for aerodynamic flows. In: 23rd fluid dynamics, plasmadynamics, and lasers conference. American Institute of Aeronautics and Astronautics, Reston, Virginia (1993). <https://doi.org/10.2514/6.1993-2906>
40. Goos, E., Burcat, A., Ruscic, B.: Extended third millennium thermodynamic database of new NASA polynomials with active thermochemical tables update (2016). Accessed 1 Sept 2022
41. ANSYS, I.: Ansys fluent user's guide, 2022R1. Ansys Fluent User's Guide (2022)
42. Mavriplis, D., Vassberg, J., Tinoco, E., Mani, M., Brodersen, O., Eisfeld, B., Wahls, R., Morrison, J., Zickuhr, T., Levy, D., Murayama, M.: Grid quality and resolution issues from the drag prediction workshop series (2008). <https://doi.org/10.2514/6.2008-930>
43. Roache, P.J.: Verification and validation in computational science and engineering. Hermosa Publications, Albuquerque, NM (1998)
44. Celik, I.B., Ghia, U., Roache, P.J.: Procedure for estimation and reporting of uncertainty due to discretization in CFD applications. ASME (2008). <https://doi.org/10.1115/1.2960953>
45. Rumsey, C.L., Smith, B., Huang, G.: Turbulence modeling resource (01.03.2022). <https://turbmodels.larc.nasa.gov/>. Accessed 20 July 2022
46. Rumsey, C.L.: Recent developments on the turbulence modeling resource website (invited) (2015). <https://doi.org/10.2514/6.2015-2927>
47. Rumsey, C.L.: Turbulence modeling verification and validation (invited) (2015). <https://doi.org/10.2514/6.2014-0201>
48. Bridges, J., Wernet, M.: Establishing consensus turbulence statistics for hot subsonic jets (2010). <https://doi.org/10.2514/6.2010-3751>
49. Bridges, J., Wernet, M.P.: The NASA subsonic jet particle image velocimetry (PIV) dataset (2011). <https://doi.org/10.13140/RG.2.1.2844.2485>
50. Rumsey, C.L.: Axisymmetric subsonic jet validation case: effect of reference Mach number (2020). https://turbmodels.larc.nasa.gov/jetsubsonic_val.html. Accessed 20 July 2022
51. Rumsey, C.L., Biedron, R.T., Thomas, J.L.: CFL3D: its history and some recent applications (1997)
52. Bush, R., Power, G., Towne, C.: WIND—the production flow solver of the NPARC Alliance (1998). <https://doi.org/10.2514/6.1998-935>
53. Mendez, M.A., Scelzo, M.T., Enache, A.: Fluidic vectoring of a planar incompressible jet flow (2018). <https://doi.org/10.1051/epjconf/201818002065>

Publisher's Note Springer Nature remains neutral with regard to jurisdictional claims in published maps and institutional affiliations.