EJECTOR TIP INJECTION SYSTEM FOR ACTIVE AERODYNAMIC COMPRESSOR STABILIZATION
PART II: CFD INVESTIGATIONS


Institute of Jet Propulsion
Universität der Bundeswehr München
D-85577 Neubiberg, Germany
sebastian.brehm@unibw.de

ABSTRACT
Injecting high-momentum jets of air into the rotor blade tip region of turbo compressors can increase its surge margin and even recover the compressor from unstable aerodynamic operation. The Institute of Jet Propulsion at the University of the German Federal Armed Forces Munich recently developed and patented a novel Ejector-based air Injection System (EIS) for such active aerodynamic stabilization of the Larzac 04 jet engine low pressure compressor. This paper deals with detailed investigations of the aerodynamic conditions within the EIS by means of CFD simulations. First, several sensitivity studies concerning the mesh resolution respectively the boundary layer treatment and the necessity to apply a transition model are conducted. The CFD predictions of the final setup are validated utilizing experimental measurements of static wall pressure distributions and the air mass flow rates involved as outlined in the first part of this paper by Kern et al. (2017). This data is also used for detailed analyses of the flow conditions within the main aerodynamic components of the EIS.

KEYWORDS
CFD, EJECTOR EFFECT, EJECTOR INJECTION SYSTEM

NOMENCLATURE

Symbols

\( \dot{m} \) [kg/s] mass flow rate
Ma [-] Mach number
\( p_w \) [Pa] static wall pressure
Re [-] Reynolds number
\( x \) [m] axial position
\( y^+ \) [-] non-dimensional wall distance
\( \mu \) [-] entrainment ratio

Subscripts
inj injection
FMOF Fully Mixed Out Flow

Abbreviations
EIS Ejector Injection System
EXP Experimental
FT Fully Turbulent mode
HPC High Pressure Compressor
IRe low-Reynolds approach
LPC Low Pressure Compressor
sWF scalable Wall Functions
TM \( \gamma-Re_\Theta \)-Transition Model

OPEN ACCESS
Downloaded from www.euroturbo.eu
Copyright © by the Authors
INTRODUCTION

One of the most challenging tasks in the design process of turbo compression systems is to achieve high efficiencies while ensuring aerodynamic stability at any time during operation. “On-demand” injection of high momentum air jets in the rotor blade tip region is a measure of active flow control promising a combination of both increased efficiencies as well as advanced operational safety. By applying such a system, a reduced stall margin is acceptable since the air injection can be activated to stabilize the compressor aerodynamics temporarily if required. This concept has been subject to numerous analytical, experimental, and numerical investigations in the past: Horn et al. (2009) for example provided a fundamental system analysis when applying tip injection for HPCs. Kefalakis and Papailiou (2006) achieved partly significant surge margin extensions both with continuous as well as pulsed injection in the tip region of a Larzac 04 LPC. Experimental as well as CFD methods have been used by Suder et al. (2000) to obtain a detailed insight into the mechanisms of active stabilization of a transonic compressor by tip air injection.

The Institute of Jet Propulsion at the University of the German Federal Armed Forces Munich has also developed such an active air injection system for stabilization of the Larzac 04 jet engine’s LPC. The system is capable of anticipating aerodynamic instabilities, to increase the surge margin of the LPC, and even to recover it from unstable operation such as rotating stall or surge (cf. Leinhos (2003), Scheidler (2005), Bindl et al. (2009), Bindl (2010)). To ensure a safe stall supression or elimination respectively, a certain mass flow rate of injected air is required. This air has to be supplied from HPC bleed ports in case of an autarkic flight application. But this recirculation of HPC bleed air is disadvantageous with respect to the overall engine efficiency due to two main reasons: First, a certain air mass flow rate already compressed by a substantial part of the compression system is diverted from the thermodynamic cycle. Second, the air injected into the compression system has an increased temperature level (cf. Kurzke (2008)). Deploying the so-called ejector effect in such an air injection system can reduce the required mass flow rate of recirculated HPC bleed air for a safe stall elimination/suppression. Subsequently, also the downgrade of the overall engine efficiency is reduced. This is achieved by blowing the HPC bleed air, called primary air, into an ejector geometry instead of injecting it directly into the tip region of the LPC. In such an ejector geometry, so-called secondary air is entrained from the surrounding atmosphere. This entrainment is caused by the momentum and energy transfer throughout the developing shear layer between injected primary and secondary air. A key parameter describing this process is the so-called entrainment ratio $\mu$. It is defined according to Eq. 1 as entrained secondary air mass flow rate divided by the primary air mass flow rate:

$$\mu = \frac{\dot{m}_{sec}}{\dot{m}_{pri}} = \frac{\dot{m}_{inj}}{\dot{m}_{pri}} - 1. \quad (1)$$

Both primary and secondary air, combining to the total injection mass flow rate $\dot{m}_{inj}$, are then injected into the tip region of the LPC. The concepts of injection and ejector injection are illustrated schematically by means of Fig. 1a and Fig. 1b respectively.

The concept of ejector injection has been patented by the Institute of Jet Propulsion at the University of the German Federal Armed Forces Munich (cf. Niehuis et al. (2012)). Stößel et al. (2014) have proven experimentally that deploying the ejector effect in the previously described manner can significantly improve impact and efficiency of the entire air injection concept. Based on these findings, a further advanced Ejector Injection System (EIS) has been developed by Kern et al. (2017) and is currently under investigation.
Besides experiments, CFD simulations are intensively used for analysis of the inner aerodynamics within the EIS. This paper is intended to validate the respective CFD setups by means of static wall pressure and air mass flow rate measurements.

Sensitivity studies concerning the numerical boundary layer treatment (low-Reynolds approach (lRe) or scalable wall functions (swF)), and the necessity to activate a transition model (TM) are conducted, too.

EJECTOR INJECTION SYSTEM

This chapter comprises only a brief overview of certain EIS features which are important for the CFD simulations of the inner aerodynamics. More details about concept, design, mechanics, and manufacturing can be found in the accompanying first part of the paper by Kern et al. (2017). Figure 2 depicts a cut through the EIS attached to the Larzac 04 jet engine with the main components outlined.

Primary air, either recirculated bleed air from the HPC (within the 4th stage) or external high-pressure air, is delivered from the primary air supply to the ejector nozzle and released from there into the mixing duct. Within the mixing duct, the ejector effect is established. The ejector effect entrains secondary air from the surrounding atmosphere via the secondary supply ducts (cf. subsection Boundary Conditions for details about the secondary air conditions).
This entrainment of secondary air is caused due to the momentum and energy transfer throughout the developing shear layer and the primary and secondary air stream start to mix out. At the end of the mixing duct, the injection nozzle is attached which guides both mixed out air streams into the tip region of the first LPC rotor. The inner aerodynamics are described further in section VALIDATION AND ANALYSES OF THE INNER AERODYNAMICS by means of measured and computed wall pressure distributions (cf. Fig. 7a and Fig. 7b).

Figure 3: Detailed arrangement of one set of supply channels between two actuators

Four of the twelve primary supply ducts in total merge from their four separate circular cross sections into a single circular ring segment as shown in Fig. 3 in red. In principal the same applies also for the secondary air supply ducts (depicted in blue in Fig. 3). However, compared to the primary air supply ducts, the change of the cross sectional areas is not as drastic. The interconnection of the two ring segments at the downstream ends of the primary and the secondary supply ducts respectively and the ejector nozzle forming the mixing duct is depicted schematically in the respective detail views in Fig. 2 and Fig. 3.

In circumferential direction, three sets of primary and secondary supply ducts (four circular connections each) are assembled between three actuators as indicated in Fig. 3. In case the EIS is attached to a jet engine, the actuators will open the injection nozzle (cf. Fig. 2) highly dynamically if required to suppress imminent aerodynamic instabilities. Thus, in total three of the ring segments mentioned above exist in the EIS, which are physically separated by the actuators. Due to the presence of the actuators and their circumferential extension, each of the three ring segments encompasses about 110° around the circumference. However, the entire EIS can be considered as periodical with respect to three segments covering a 120° sector in circumferential direction (120°-segment indicated in yellow in Fig. 3).

CFD SETUP

Domain extension

As already described previously and depicted in Fig. 3, the entire EIS can be considered as periodical with respect to three segments covering a 120° sector each. This periodicity is exploited by abstraction of the simulation domain from the full annulus EIS into such a segment marked in Fig. 3. It has to be emphasized that this kind of abstraction is not causing any simplifications apart from minor geometry modifications such as reduction of bolt heads for example.
Although the number of primary and secondary air supply ducts is equal, a 120° sector has to be simulated if the wake regions downstream of the actuators are to be considered. Thus, a geometrical scaling of the one primary and one secondary supply duct to e.g. 30° circumferential extension is no option.

The primary air supply ducts are extended further upstream by cylinders to emulate the boundary layer built-up when the air supply hoses are connected within the experiments (cf. Fig. 4a in red). The axial lengths of these have shown practically no influence in preliminary sensitivity studies. The entire domain extension can be seen in Fig. 4a and Fig. 4b.

**Computational meshes**

Due to the highly complex geometries involved, the meshing of the entire domain is split up in several independently meshed modules interconnected by so-called “CFX General Grid Interfaces”. These are interpolating the respective transfer parameters at the mesh module interfaces for non-matching nodes (details are given in ANSYS Inc. (2011)). By means of this modular approach, the topology and mesh characteristics can be optimized for the geometry of the respective module. Thus, a single grid for the entire EIS can be avoided which could not be generated time efficiently due to the highly complex geometry. The entire domain extension as shown in Fig. 4a consists of nine single mesh modules in total (each one in different colors; four individual primary air supply ducts depicted in red in Fig. 4a). Tetrahedral meshes with prismatic wall layers are used for the primary air supply and intermediate module 1 (depicted as purple and blue in Fig. 4a). The remaining modules are meshed with hexahedral cells. First, meshes are created according to the existing best-practice knowledge targeting on $y^+ \approx 30$ at all no-slip walls to make use of (scalable) wall-functions for the boundary layer treatment. This is resulting in a total node number of 11,593,801. These meshes are taken as basis and refined to yield $y^+ \approx 1$ to enable for a low-Reynolds approach. The total node number of 21,853,050 is result of subsequent mesh sensitivity investigations which are not shown here for comprehensive reasons (cf. also subsection Entrainment ratio). Fig. 5 shows exemplarily the computational mesh within a cross section throughout the region at which the first interaction and mixing between primary and secondary air mass flow rate takes place and which is therefore also the most important part of the domain from an aerodynamical and numerical point of view.

![Modular meshing concept](image1)

![Boundary conditions](image2)

**Figure 4:** EIS CFD setup
Boundary conditions

Within the presented paper, the EIS is operated in so-called "stand-alone mode" only. This means that it is not attached to the jet engine and pressurized air from a screw compressor is used as primary air. The stand-alone mode is used within series of extensive pre-investigations on the EIS prior to attaching it to the Larzac 04 (cf. also the first part of the paper for details; Kern et al. (2017)). Simulations which also incorporate the impact of the engine mass flow rate (engine test facility operation) and/or the stagnation conditions at the secondary air and intake inlet due to the aircraft speed in an autarkic flight application are intended for future publications. It is expected from analyses already conducted that the secondary air mass flow rate can increase partly significantly for both mentioned types of operation compared to stand-alone mode. The boundary conditions applied to simulate stand-alone mode are depicted in Fig. 4b. Mass flow rate and total temperature of the primary air (both measured) are specified at the inlets of the primary air supply ducts (indicated in red in Fig. 4b). A uniform distribution is chosen as the boundary layers are emulated by the extension. Variation of the extension’s length has shown a diminishing influence in preliminary sensitivity analyses, as already described previously in subsection Domain extension. Atmospheric pressure and temperature are set at the inlets of the secondary air supply duct (turquoise in Fig. 4b) as well as at the in- and outlet of the intake (purple and yellow respectively in Fig. 4b). Thus, steady atmospheric conditions are modeled there. As the air is aspirated directly from the atmosphere into the secondary air inlet and the intake inlet, uniform distributions of total pressure, temperature, and turbulence are specified. Turbulence intensities are set to 5% at all inlets (initial start for future parameter studies). The flow at the intake outlet is modeled as “adapted nozzle flow” into the surrounding atmosphere as the ending of the EIS casing in “stand-alone mode” is coincident with the location of the boundary condition and therefore the pressure distribution is set as uniform there. All walls indicated in gray in Fig. 4b are specified as no-slip (hydraulically smooth) and adiabatic. Symmetry boundary conditions are applied for both lateral faces which result from the domain abstraction (shown in green in Fig. 4b).
Numerical settings

All simulations described within this paper are carried out in RANS-mode of the commercial solver ANSYS CFX 14.0. The two equation shear stress transport (SST) k-ω-turbulence model is utilized. It is either used in Fully Turbulent mode (FT) or combined with the γ-ReΘ-Transition Model (TM). CFX applies scalable Wall Functions (sWF) for the grids with $y^+ \approx 30$. These do not have to be applied in case of the low-Reynolds-approach (lRe) with the meshes providing $y^+ \approx 1$. The wall boundary layers are fully resolved in these cases according to ANSYS Inc. (2011).

VALIDATION AND ANALYSES OF THE INNER AERODYNAMICS

Measurement equipment

This chapter will only contain basic information required for the validation of the CFD results. Details about selection, installation, and data acquisition of the measurement equipment are given in the accompanying first part of the paper by Kern et al. (2017). The primary air mass flow rate is measured by means of a coriolis sensor installed in one of the twelve air supply hoses which achieves a nominal measurement uncertainty of 0.25% for most operating points. The coriolis sensor also incorporates a PT100 sensor for determination of the primary air’s recovery temperature. As outlined in detail in Kern et al. (2017), the secondary air mass flow rate can be determined in two different ways: First, a hybrid system consisting of a two-path ultra sonic tube and pneumatic measurements within a bellmouth. This system can be attached with connection hoses to the secondary air supply ducts. Second, pneumatic measurements directly within one secondary supply duct can be utilized which applies for the verification data used within this paper. Thirtyfive static wall pressure taps installed within the outer walls of the primary air supply duct (5 out of 35), the ejector nozzle (2), the mixing duct (21), and the injection nozzle (7) are also used for validation. These pressure taps are aligned in axial direction at the same circumferential position (symmetry plane of one primary/secondary duct).

Figure 6: Dependencies on primary air mass flow rate
Entrainment ratio

As described above, the entrainment ratio $\mu$ is the key parameter with respect to the EIS aerodynamics. Subsequently, the sensitivity analyses concerning mesh resolution and numerical boundary layer treatment are limited to $\mu$ within this publication for comprehensive reasons. Figure 6a depicts the entrainment ratios $\mu$ measured experimentally and predicted by three different CFD setups for different primary air mass flow rates: Meshes for application of sWF ($y^+ \approx 30$) in combination with fully turbulent simulations (CFD-120°/sWF/FT), meshes enabling a low-Reynolds approach ($y^+ \approx 1$) combined with fully turbulent simulations (CFD-120°/lRe/FT), and meshes enabling lRe with activated transition model (CFD-120°/lRe/TM).

It can be seen that the best fit with the experimental measurements is obtained with CFD-120°/lRe/TM as the quantitative differences range from only from about $\approx 3.6\%$ to about $\approx 7\%$. Although, it can also be recognized that the maximum of the entrainment ratio is predicted by CFD for lower primary air mass flow rates compared to $\mu_{\text{EXP}, \text{max}}$. The CFD-120°/lRe/TM-setup requires the highest computational effort due to two additional transport equations in the presented case and an almost doubled number of nodes compared to sWF simulations (result of mesh independency studies not shown here due to comprehensive reasons as described above). However, additional effort for a transition model is required as CFD-120°/lRe/FT shows up to $\approx 17\%$ lower entrainment ratios compared to the measurements. An even lower entrainment is computed by fully turbulent CFD simulations with sWF applied. In addition to that also the qualitative prediction of the coherence between primary air mass flow rate and $\mu$ is insufficient as the entrainment is steadily increasing with increasing primary mass flow rate.

Thus, the following conclusion can be drawn: Fully resolving the boundary layers with meshes allowing for a low-Reynolds approach and including transition modeling is recommended to obtain best fit with the data determined experimentally. The main reason therefore is believed to originate from the computation of the flow within the ejector nozzle (cf. detail view of Fig. 2 and Fig. 5) as the most apparent differences between CFD-120°/lRe/FT and CFD-120°/lRe/TM can be found there. Within the ejector nozzle, the computation of the numerical intermittency suggests a laminar flow regime for CFD-120°/lRe/TM even at the maximum primary mass flow rate. The shapes of the flow profiles across the ejector nozzle support this and point at re-laminarization within the slim ejector nozzle (smaller than a single digit millimeter). This has two major consequences on the flow conditions within the primary nozzle exit cross section (indicated in Fig. 2 by means of a black line) which is the most important position within the EIS because instantaneously downstream, the interaction between primary and secondary air starts: First, as the laminar regime is causing a lower skin friction leading to lower total pressure losses, the total pressure level at the primary nozzle exit cross section is increased. Second, also the blockage impact of the boundary layer profiles is increased which means to allow for the same primary air mass flow rate $\dot{m}_{\text{pri}}$, the flow must be accelerated further resulting in higher peak Mach numbers. This point is also discussed in detail within subsection Wall pressure distributions. However, which of the mentioned factors concerning the flow conditions within the primary nozzle exit cross section are of major and which are of minor impact on the predictive capabilities of the ejector effect dominated aerodynamics has to remain to be subject of current and future research. This is due to the extraordinary computational effort large scale parameter studies obligatory therefore are requiring. In order to investigate the aerodynamic conditions within the primary nozzle exit cross section further, the development of $M_{\text{a,pri}}$, $Re_{\text{pri}}$ for CFD-120°/lRe/TM depending on the primary air mass flow rate is depicted in Fig. 6b.
All fluid mechanic parameters are mass flow averaged over the entire nozzle exit cross section and the hydraulic diameter of the cone envelope segment shaped exit cross section \(d_{hyd}\) is used for determination of \(\overline{Re}_{pri}\). The Mach number \(\overline{Ma}_{pri}\) is linearly increasing with increasing primary air mass flow rate. It can be seen that the primary nozzle exit cross section is not chocked yet with the maximal primary mass flow rates simulated as \(\overline{Ma}_{pri,max} \approx 0.6\). CFD simulations of a basic ejector effect test case conducted by Brehm and Niehuis (2015) suggest that the entrainment is decreasing with increasing primary air mass flow Mach number (for equivalent \(\overline{Re}_{pri}\)). However, also the primary mass flow Reynolds numbers shows a linear dependency on \(\dot{m}_{pri}\). Brehm and Niehuis (2015) also showed by means of CFD simulations of a basic ejector test case that the entrainment increases with increasing \(\overline{Re}_{pri}\) up to about \(\overline{Re}_{pri} \approx 10^6\). Beyond \(\overline{Re}_{pri} \approx 10^6\), the differences are less than 1% for further increasing Reynolds numbers. As \(\overline{Re}_{pri,max}\) is only about 22,500 for the EIS and thus two orders of magnitude lower, a strong mutual interdependency of both the effects of varying \(\overline{Ma}_{pri}\) and \(\overline{Re}_{pri}\) on the entrainment ratio is suspected.

In addition to that, Fig. 6b provides also information about the state of mixing between the primary and the entrained secondary air mass flow within the mixing duct of the EIS by means of \(\frac{x_{FMOF}}{x_{mix,max}}\). Thereby, \(x_{FMOF}\) describes the axial position within the mixing duct at which both air streams are fully mixed out. The criterion to determine a fully mixed out flow is adopted from Brehm and Niehuis (2015): If velocity profiles in radial direction throughout the mixing duct do not exhibit a point of inflection any more, the flow is considered as fully mixed out up from this position. Thus, to determine \(x_{FMOF}\), a fully automated post-processing script extracts velocity profiles in radial direction at numerous axial positions within the mixing duct. These are analyzed for points of inflection by an implemented numerical second order differentiation scheme subsequently. As the mixing duct geometry is fixed but the momentum of the primary air mass flow is increasing with increasing primary mass flow rate, \(\frac{x_{FMOF}}{x_{mix,max}}\) shifts downstream (cf. Fig. 6b). The basic ejector research mentioned (Brehm and Niehuis (2015)) has also shown that a maximum of the entrainment ratio exists in cases where the velocity profiles are fully mixed out within the mixing duct exit plane independently of all other parameters investigated. Thus, also an influence of the mixing characteristics is assumed in the presented case of the EIS.

It has been shown that several aerodynamic parameters are suspected to have simultaneous (and most likely interdepending) influence on the entrainment ratio. Large scale parametric CFD simulations are required for a further analysis of the decoupled quantitative contribution of each parameter to the coherence between primary air mass flow rate and entrainment ratio and also the predictive capabilities (cf. section CONCLUSION AND OUTLOOK).

Wall pressure distributions

In addition to the validation by means of the global parameter entrainment ratio \(\mu\), also local static wall pressure distributions measured and predicted by CFD are compared. In Fig. 7a the static wall pressures normalized with atmospheric pressure at the upper casing of the primary duct, the ejector nozzle, the mixing duct, and the injection nozzle is shown for the lowest primary air mass flow rate \(\dot{m}_{pri}/\dot{m}_{pri,max} = 0.08\). A Mach number contour plot on the symmetry plane of one of the four supply ducts with a detail view is also included for sake of clarity.
A slight acceleration of the flow can be noticed around position I where the cylindrical extension of the primary air supply is connected to the primary duct. Between position II and III, the flow is strongly accelerated since the cross sectional areas of the primary duct are decreasing. The same applies also for the ejector nozzle (position IV to V). The highest velocities (suction peak) occur around V, coincident with the ending of the Coanda radius between ejector nozzle over which the flow is accelerated (cf. detail view of Fig. 7a). Downstream of position V, the alternation of the wall pressure around the pressure level of the surrounding atmosphere itself remains relatively small. This shows that the inner aerodynamics within the mixing duct and the injection nozzle are mainly dominated by the conditions in which primary and injection mass flow rate are "released". Within the remaining part of the mixing duct, a continuous momentum and energy transfer from the primary to the secondary air takes place. This causes a gradual deceleration of the relatively fast primary air jet at the upper side of the mixing duct from V to VII. Subsequently, the wall pressures are gradually increasing. This deceleration is superimposed by two additional effects around VII and VIII respectively: First, a contraction of the cross-sectional area leads to a local acceleration at VII when the flow approaches the injection nozzle. As the transversable injection nozzle spacer (cf. Fig. 2) is fully open within the simulations presented here, the decrease in static wall pressure remains relatively low. Second, a local acceleration of the flow occurs along the convex curvature of the injection Coanda radius connecting the upper wall of the injection nozzle and the inner side of the intake (position VIII).
The wall pressures predicted by CFD are less than 0.1% lower compared to the experiments between position I and V for $\dot{m}_{pri}/\dot{m}_{pri,\text{max}} = 0.08$. Downstream of position V, the differences between experiments and CFD-120° are marginal and thus $p_{w,\text{CFD-120°/lRe/TM}}$ and $p_{w,\text{EXP}}$ are practically identical. It can be concluded that the developed CFD setup shows a very good prediction in this case.

The static pressure distributions along the upper walls are qualitatively almost similar for all primary air mass flow rates investigated. The only exception is between position III and IV where a "pressure plateau" is established for higher primary mass flow rates which becomes the more pronounced, the higher $\dot{m}_{pri}/\dot{m}_{pri,\text{max}}$. This is exemplarily demonstrated by means of Fig. 7b where the wall pressure is depicted for the highest primary mass flow rate investigated. It can be seen that the quantitative pressure levels are shifting and that the deviations between EXP and CFD-120°/lRe/TM are increasing with higher primary air mass flow rates. They reach about 8% for $\Delta p_{w,\text{CFD-EXP}}$ between position I and II and an even increased offset can be noticed for the axially aligned part of the ejector nozzle (between III and IV): There, $p_{w,\text{CFD}}$ is lower by about 9% compared to the measurements at $\dot{m}_{pri,\text{max}}$. But the wall pressures measured and predicted by CFD-120° are practically identical downstream of position V analog to $\dot{m}_{pri}/\dot{m}_{pri,\text{max}} = 0.08$. Thus, deviations are only present within the primary air supply duct and the ejector nozzle but not in the region where the momentum and energy transfer from the primary to the secondary flow takes place. The differences might be explainable by the computation of the boundary layer profiles within these components by the CFD solver: The boundary layer displacement thicknesses seem to be overpredicted and thus the core flow is accelerated further compared to the experiments resulting in lower static wall pressures in CFD. This effect is more pronounced if the part of the cross sectional area influenced by viscous effects becomes larger which is the case for the ejector nozzle (position III to V; cf. also subsection Entrainment ratio). The more intense acceleration in CFD is most likely also resulting in higher $\overline{M_\text{pri}}$ and $\overline{Re_\text{pri}}$ within CFD. Above, the influence of these parameters on $\mu$ has been discussed and this seems to be also a reason for the deviations present between CFD and experiments (cf. Fig. 6a).

Ultimate confirmation of the suspicions outlined would only be possible by means of boundary layer measurements within the primary air supply duct. This is currently not feasible due to the small dimensions and the absence of any optical access. However, this topic remains subject of current CFD research at the institute to improve the predictive capabilities beyond the presented state (cf. section CONCLUSIONS AND OUTLOOK) although large scale parametric studies are hardly feasible due to the extraordinary computational effort.

CONCLUSION AND OUTLOOK

Within this paper, CFD analyses of an Ejector Injection System (EIS; cf. Fig. 2), developed and patented by the Institute of Jet Propulsion (cf. Niehuis et al. (2012)) for active aerodynamic stabilization of turbo compressors are presented. First, sensitivity studies concerning the CFD setup in ANSYS CFX, which exploits the periodicity of the EIS in circumferential direction (120° segments), are conducted. By means of experimentally determined ejector entrainment ratios for different primary air mass flow rates it can be stated that it is necessary to resolve all no-slip wall boundary conditions with a low-Reynolds approach and to apply a transition model.
The additional computational requirements (node numbers almost doubled and two additional transport equations) have to be accepted since this setup shows qualitatively and quantitatively the best agreement with the experiments. The reasons therefore are believed to originate from a different computation of the flow profiles within the exit cross section of the ejector nozzle. There, the simulations with utilized transition modeling suggest laminar flow regime which remains subject to intensive current and future research. Deviations between $\mu_{\text{EXP}}$ and $\mu_{\text{CFD-120}^\circ/\text{Re}/\text{TM}}$ are still present but remain moderate with a maximal difference $\approx 7\%$.

It has also been shown that three aerodynamic parameters, the Mach and Reynolds number of the flow within the exit cross section of the ejector nozzle $Ma_{\text{pri}}$ and $Re_{\text{pri}}$, as well as the mixing state of the primary and the secondary flow are simultaneously changing with varying primary air mass flow rate and are strongly linked amongst each other. All these parameters have shown to exert influence on the entrainment ratio revealing the necessity of further investigations to determine the quantitative contributions of all these parameters.

Second, the results obtained with the CFD-120$^\circ/\text{Re}/\text{TM}$-setup are validated in detail by means of measured static wall pressure distributions, which are predicted qualitatively very well. Quantitative deviations only occur within the primary supply ducts and the ejector nozzle. These increase from less than 0.1% for the lowest primary air mass flow rate simulated to about 9% for $\dot{m}_{\text{pri}}/\dot{m}_{\text{pri,max}} = 1$. Within the mixing duct and the injection nozzle on the other hand, there are practically no differences noticeable between CFD predictions and measurements. It can also be stated that the flow conditions within these EIS components are mainly governed by the pressure level in which the primary air mass flow is released.

It has been shown that the developed CFD setup is capable of predicting both global and local features of the EIS inner aerodynamic which applies especially for low primary air mass flow rates. The quantitative deviations for entrainment ratio and wall pressure distributions at higher primary air mass flow rates elaborated within this paper are addressed in further research activities at the Institute of Jet Propulsion focusing on:

1. Additional investigations are planned on the boundary layer development within the primary and secondary air supply ducts and the ejector nozzle including large scale variation of turbulence settings and turbulence models. This is rather complicated since these variations are suspected to have an impact on the computation of the momentum and energy exchange throughout the shear layer too, which in turn has direct impact on the predicted entrainment ratios.

2. It is also intended to verify an approach for exploitation of the mainly present axisymmetry of the EIS by abstraction into a fully axisymmetric domain. This would enable a significant reduction of the computational requirements which allows for extremely efficient parametric CFD studies. This in turn is especially favourable for a decoupled variation of primary air Mach and Reynolds number $Ma_{\text{pri}}$ and $Re_{\text{pri}}$, the state of mixing between primary and secondary air, turbulence settings and models, as well as different shapes of the flow and boundary layer profiles within the ejector nozzle exit cross section as addressed above.
REFERENCES

ANSYS Inc. ANSYS CFX, Release 14.0, 2011.


