ISABE-2015-20019

Aspects of Serpentine Inlets for Integrated Propulsion Systems

Stefan Bindl, Rudolf P.M. Rademakers, Sebastian Brehm, Reinhard Niehuis
Institute of Jet Propulsion
University of the German Federal Armed Forces
85579 Neubiberg, Germany
E-Mail: stefan.bindl@unibw.de

ABSTRACT

For the operation of gas turbine engines, the inlet plays a major role by providing the air necessary for the engine's thermodynamic process. Especially the flow conditions and hence the pressure losses have to be taken into account while designing such inlets. In most civil aircraft conventional inlet geometries with axial through flow are used. The only exceptions are some triple engine configurations. However, future aircraft designs, like the blended wing body concept, often feature integrated propulsion systems in order to reduce drag. Furthermore, there are requirements in some modern military air systems that cannot be covered by conventional inlet geometries.

The inlet system is traditionally designed by the aircraft manufacturer and the flow field is given to the engine manufacturer by distortion parameters. This subdivision leads to a number of disadvantages or at least to a waste of efficiency potential. For this reason, a number of initiatives have been started to bring both sides together combining the inlet and the engine to a propulsion system approach. Within this paper the most significant requirements for modern serpentine inlet geometries, used in those propulsion systems, are addressed and discussed. Furthermore, relevant description parameters are assessed and some measurement rake designs are evaluated concerning their capability to determine the relevant data.

NOMENCLATURE

Symbols

Θ	[°]	circumferential extent
Π	[-]	pressure ratio
c	[m/s]	velocity
L	[m]	length of center line
n	[%]	relative spool-speed
p_t	[Pa]	total pressure
x	[m]	coordinate along center line
PAV	[Pa]	averaged total pressure
PAVLOW	[Pa]	mean total pressure below PAV
PFAV	[Pa]	face-average total pressure

Abbreviations

AIP	Aerodynamic Interface Plane
CDI	Circumferential Distortion Intensity
CFD	Computational Fluid Dynamics
DC(Θ)	Distortion Coefficient
IR	InfraRed
MPS	Maximum Power Setting
PIV	Particle Image Velocimetry
RCS	Radar Cross Section
RDI	Radial Distortion Intensity
SC(Θ)	Swirl Coefficient
SFC	Specific Fuel Consumption
UAS	Unmanned Aerial System

INTRODUCTION

During the next years various aircraft manufacturers will offer successors for their well established products. This next generation of commercial airliners will stick to the approved design with up to date engines (NEO) mounted in a conventional way. Although there are airliners, like the Boeing 727 or the Lockheed L1011 Tristar, featuring curved inlets for the number 2 engine, the intakes only have moderate shapes with two bends and a fairly straight portion in the middle. In case of the L1011 the engine delivers performance equivalent to the wing mounted engines.

For the kind of inlets used in these aircraft many studies have been performed in the past especially focused on flow separation and secondary flow fields inside the ducts [1,2]. For a general understanding of these emerging flow mechanisms those investigations give a well-founded basis. In contrast to the above mentioned configurations future military UASs will feature fully integrated propulsion systems. Signature control is thereby the main challenge which makes it necessary to hide the engine inside the airframe. Besides the IR signature which is basically influenced by the nozzle design, the air intake is a key factor for radar signature. While the fuselage can be designed with a smooth surface, the fan stage of the engine can not be adapted. For that reason the only way to reduce RCS of this component is to hide it behind an s-shaped inlet duct with no direct line of sight. Avoiding radar beams reaching the fan blades, the contour of the duct has to be much more aggressive than those of commercial airliners. Due to flow separation inside of such highly bent inlets, the formation of vortices and therefore a combination of total pressure and swirl distortion is induced. Hence, the aircraft engine is prone to those inhomogeneous inlet conditions with negative influence on thrust, efficiency, and the stability of the compression system. Distortions in the pressure field can even lead to fan and compressor blade fatigue and often also increase noise [3].

As it can clearly be derived from the stated points there is a strong interaction between the inlet duct and the engine. Hence, new and enhanced techniques are necessary for understanding inlet-engine-integration effects in order to meet future requirements [4]. At the Institute of Jet Propulsion intensive efforts are made to achieve this goal by combining experimental investigations at turbofan engines with high performance CFD simulations.

SERPENTINE INLET DUCTS

Requirements for modern s-ducts

The air intake for a gas turbine engine is a crucial part of the propulsion system as it directly interacts with the internal airflow to the engine and hence affects its performance characteristics.

As described previously there are many applications for s-ducts. Depending on the type of aircraft and the design of the propulsion system, the requirements for those inlet ducts can differ. Especially in military aviation there are several tasks to be covered by the inlet besides the sufficient supply of air to the jet engine [5].

- Signature Control
  Today's manned fighter aircraft as well as unmanned aerial systems feature stealth technology. Signature reduction is one of the key factors to enhance survivability by avoiding adversary offensive action.
  If designed properly the inlet duct can help to reduce radar as well as noise emission of the jet engine. The s-shaped design with no direct line of sight to the fan stage of the engine keeps the radar reflections to a minimum and makes it almost impossible to identify the engine and therewith the aircraft. In order to further reduce the signature, the air inlet can be internally covered with radar absorbent materials or the installation of additional radar blockers.
  Another relevant task is the reduction of noise. In particular with gas turbine propulsion this issue gains in importance. Besides the jet noise, which has to be controlled by the nozzle, the inlet can cover the fan noise from forward propagation. Hence, the acoustic signature of an approaching UAS can be limited. On the other hand distortions in the pressure field as a result of the curved inlet can cause increased fan noise.

- Flight Condition Independence
  Whether the aircraft is on the ground, at very low altitudes or flight at high altitudes the engine inlet has to be able to provide high-quality airflow to the engine. Additionally a wide spectrum of speeds and maneuvering conditions have to be covered by the inlet. Especially configurations with narrow intakes which in most cases are integrated into the shape of the fuselage are prone to flow separation at high angles of attack or cross winds. Thus, the proper operation of the inlet under all the very different external conditions has to be assured. Furthermore, it has to provide an appropriate amount of airflow to the engine from idle to maximum military or even afterburning power in order to fulfill the mission.

- Fuselage Integration
  As already mentioned, the air intake

has to be highly integrated into the fuselage of the aerial vehicle to realize stealth capabilities. If integrated into the wings the engine intakes tend to disturb the laminar flows and therefore the good aerodynamic performance. The inlet cross sectional area is thereby given by the engine and only the shape can be varied to fit the wings/fuselages contour. Additionally the requirements necessary for sufficient flow quality inside the duct have to be taken into account. Therefore, the inlet is mostly a tradeoff between those tasks. The use of CFD-methods improved the designs during the last years. Even the avoidance of boundary layer separators could be achieved for the latest generations of UAS by diverterless inlets.

Flow conditions and secondary flows

The flow conditions within contoured air inlets are dominated by secondary flow phenomena. Thereby the origin of separation and swirl is located in different areas.
A flow separation from the walls of the inlet duct emerges when velocity and convexity are too high for the fluid to follow the contour. In that case the flow detaches and recirculation areas can establish. In consequence the aerodynamic cross sectional area is reduced. Especially lip separation can cause severe blockage of the inlet area and therefore high velocities or reduced mass flow rates at off design conditions.

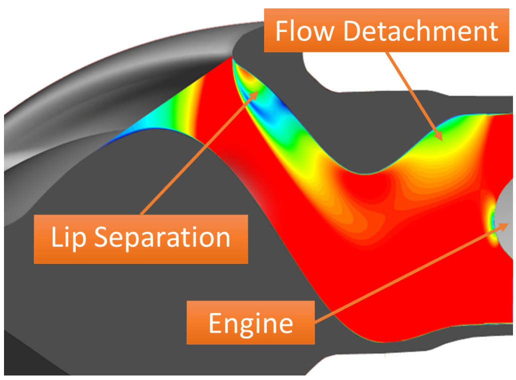

Figure 1: Separation effects

Due to the bends and thus the pressure gradient between the inner and outer walls, resulting from the centrifugal forces, the boundary layer from the outside of the first bend moves towards the inside. This results in some amount of swirl propagating further downstream. For excessively steep bends the swirl distortion increases initiated by flow detachment from the walls. In addition lip separation can cause swirl effects at the far beginning of the duct that have a severe influence on the flow conditions further downstream. In direct consequence to the swirl distortion the jet engine can suffer thrust losses up to 10% as energy is lost to accelerate the flow in angular direction [6].
For twin engine applications the problem resulting from inlet swirl is mostly limited to one of the engines. When generated in the same direction of compressor rotation, bulk swirl can even enhance compressor stability, while swirl rotating against compressor rotation adversely affects the fan stability.
Along with the mentioned flow effects a loss of total pressure is associated. This always results in an alteration of the operating conditions of the turbo compressor and therefore the entire engine performance. Due to the total pressure loss the thermodynamic process is less efficient and in consequence higher SFC or thrust reduction are caused. In consequence, the design of the inlet is of great importance for the internal flows and hence for the engine performance.

Design and optimization

Historically the design of the intake was under the responsibility of the aircraft manufacturer. However, to ensure the safe operation of the engine the results of wind tunnel tests have been evaluated by certain parameters and given to the engine manufacturer. The increasing requirements for inlet ducts results in a closer look on coupling effects between engine and inlet. In future only the design of the propulsion system in total can guarantee the best solutions [4].
Some of the rudimentary requirements for a jet engine inlet, besides signature control and others, are summarized by Zelenak [7]:
The inlet

- must supply the amount of air required for proper engine operation
- contributes to total aircraft drag in a minimal way
- diffuser should reduce the velocity of incoming air in an efficient manner

- has as small a loss in total pressure as possible
- provides uniform airflow to the engine

For highly bent inlet ducts especially the last two points are challenging to achieve. As described by Ramachandra et al. [8] inlets with high average pressure recovery values are often sensitive to off-design operation.
In order to not only reduce total pressure losses but to realize a good uniformity of the flow, the cross-section and center line design are of great importance. Parameters like entry-exit-plane-offset and length dramatically influence the flow conditions at the AIP. Furthermore, high variation in cross-sectional area or disadvantageous contours abet the formation of secondary flow phenomena. Especially the occasional high difference between the geometrical and the aerodynamic area in some sections has to be taken into account during the design process.
How an automated design optimization for a given set of boundary conditions can be realized was demonstrated by Siller and Voß [9]. A genetic algorithm was used in combination with CFD finding an optimal solution considering vertical offset and flow quality at the aerodynamic interface plane.

DESCRIPTION PARAMETERS

For the quantification of the inlet distortion within the AIP the engine manufacturers use different distortion parameters. Some of them have been established and are commonly used by the community. These parameters can be subdivided into ones describing the pressure distortion as well as those evaluating the swirl pattern. All of them are evaluated at the point where the duct ends and the engine begins namely the aerodynamic interface plane.

- Inlet Pressure Recovery
  One of the most important parameters related to the entire turbo machine is the total pressure loss. In order to keep efficiency high the losses of total pressure have to be minimized. This is in particular evident for the inlet duct. While in conventional cowlings the pressure loss is very low, bent ducts generate quite high losses due to the secondary flow effects. A possible parameter to describe the quality of the inlet related to total pressure loss is the inlet pressure recovery (IPR).

$$\Pi_{rec} = \frac{\overline{p}_{t,AIP}}{p_{t,\infty}}$$

- Distortion Coefficient
  Local pressure gradients have negative effects on compressor stability. However, if these distorted sectors are small enough the compressor will not respond with reduced surge margin (critical angle concept, see [10]). Furthermore, a distortion with large circumferential extent causes an almost homogeneous flow field in front of the compressor and therefore has a limited influence on stability as well.
  Back in the 1960s tests at Rolls-Royce by Seddon and Goldsmith [11] revealed the distortion covering a sector of around 60 degrees as worst for the compressor. In consequence the Distortion Coefficient (DC(Θ)) was introduced which is commonly used with the proposed 60 degree segment as DC60

$$DC(\Theta) = \frac{\overline{p}_{t,AIP} - \overline{p}_{t,\Theta}}{\overline{p}_{t,AIP} - \overline{p}_{AIP}}$$

  The DC60, which is a special form of CDI (see next section), is often used since for engine certification.

- Circumferential Distortion Intensity
  The circumferential distortion intensity (CDI) is evaluated within 'i' rings of a measurement rake in which probes have an equal position in radial direction.

$$CDI_i = \frac{PAV_i - PAVLOW_i}{PAV_i}$$

  The average pressure PAV is calculated as the ring averaged total pressure whereas PAVLOW is the mean value of the total pressure values below PAV within each ring.

$$PAV_i = \frac{1}{360} \int_0^{360} p_t(\Theta)_i \, d\Theta$$

$$PAVLOW_i = \frac{1}{\overline{\Theta_i}} \int_{\overline{\Theta_i}} p_t(\Theta)_i \, d\Theta$$

Either the mean or the maximum of i-values can be used for analysis and correspondingly indicated as CDI_{mean} or CDI_{max}, respectively.

- Radial Distortion Intensity
  According to the CDI the radial distortion intensity (RDI) is defined as the difference between the area-weighted face-averaged total pressure and the average total pressure, divided by the first.

$$RDI_i = \frac{PFAV - PAV_i}{PFAV}$$

- Swirl Coefficient
  Swirl distortions generated by bent inlet ducts are often characterized by means of the so-called SC(Θ) coefficient. It relates the absolute maximum circumferential cross flow velocity in an arc segment subtending Θ (commonly 60°) $|\vec{c}_{\Phi,max,\Theta}|$ to the mean throat velocity at the AIP $|\overline{\vec{c}_{AIP}}|$.

$$SC(\Theta) = \frac{|\vec{c}_{\Phi,max,\Theta}|}{|\overline{\vec{c}_{AIP}}|}$$

If both pressure and swirl distortion should be taken into account and be described by only one parameter, an offset is commonly added to e.g. the DC60 parameter to cover the additional swirl distortion. Indeed, an uncertainty and a reduced transferability results from this approach as different engines are more or less prone to swirl. It has to be taken into account as well that different kinds of swirl have different influence on compressor stability. The described procedure works well for bulk swirl but becomes problematic for inlets generating much more complex swirl patterns. For this reason an evaluation clearly distinguishing between pressure and swirl distortion parameters will lead to more accurate stability predictions for the engine.

CHALLENGES IN SIMULATION AND EXPERIMENT

In order to predict inlet distortion for new aircraft concepts or to evaluate engine response to given geometries, the flow simulation or wind tunnel testing of inlet ducts plays a major role. Lots of investigations concerning this topic has been done in the past, but with more advanced future inlet geometries new challenges in simulation and experiment arise.

Issues for simulating duct flows

The availability of high performance computer resources more and more enable the use of CFD-methods for the design of inlet ducts. Especially advanced designs for fighter aircraft or UAS profit from these developments. On the other hand not only the resources but the simulation tools are responsible for realistic results. Hence the simulation quality, which is mainly determined by realistically capturing the flow separation, is a key factor and depends mainly on

- boundary conditions,
- turbulence modeling, and
- secondary flow phenomena.

It is well known that the compressor feeds energy upstream into the duct flow. Especially while prone to inlet distortion these effects get highly inhomogeneous. The interactions between both components mainly alter the local mass flow distribution and therefore the pressure field at the duct exit of the duct. In consequence the separation inside the duct is weakened [12,13]. Assuming that the boundary conditions are set at this interface the interactions have to be covered by the simulation in order to generate realistic results. Another approach is to jointly compute the inlet and the compressor in combination using two different specialized solvers. This approach was pursued by Niehuis et al. [14]. The TAU code (specialized for outer aerodynamics) was thereby used for simulating the distortion generated by the inlet while the TRACE code is especially developed for the turbo component simulation. The challenging task with this is to realize an interface between the two simulation methods. In a first step the TRACE code was enabled to compute not only the passage flow within the compressor stage but also the duct flows. First attempts show promising results enabling a proper simulation of the inlet-compressor-interaction. Comparable work was done by Hale et al. [15] who analyzed a multistage compressor together with inlet distortion using TEACC (Turbine Engine Analysis Compressor Code). Even if a simulation setup works properly the question arises if these calculations have to be performed in unsteady mode to capture time dependent phenomena. In consequence the computation efforts are increased by far and the use for non-academic tasks becomes questionable.

Measuring flow distortions

While already challenging on rig installations, adequate instrumentation of flight test vehicles is crucial for gathering sufficient data at the interface plane. For all the different kinds of test setups diverse instrumentations are available. The optimal design of a measurement rake depends on both the sort of test and the kind of distortion, which has to be measured.
Commonly two kinds of rakes can be distinguished: traversable rakes and non-displaceable rakes.

- Traversable rakes
  This kind of rake comprises just one or a few probes which are moved within the measurement plane to achieve sufficient data resolution. While the traversing is in progress the operating conditions have to maintain stable. Hence, this instrumentation is often applied to rigs and engine test beds. The minor costs due to the reduced number of probes is all or partially diminished by the expenses for traversing equipment and longer test duration. But the number of measurement points can be arbitrary. For stationary distortion a very high resolution can be realized while dynamic distortion cannot be gathered within the entire AIP at once. Today the very stable operation of engines due to spool-speed control regimes enables even long measurement campaigns.

- Non-displaceable rakes
  In cases where a traversable rake cannot be installed, e.g. flight testing, non-displaceable rakes are used. Commonly these rakes comprise struts in length of the inlet diameter due to structural integrity. The number of arms mainly depends on the measurement resolution to be achieved and the blockage caused by the installation.
  In ARP-1420 [16] the use of a rake with 8 arms and 5 probes each is suggested. This setup dates back to inlet investigations related to fighter inlets with a high length-to-diameter ratio. To what extent this can be transferred to today's highly bent s-shaped inlet configurations is at least questionable. Depending on the expected distortion, data resolution, and blockage the use of a higher number of arms is reasonable.
  Some studies have been conducted in the past to evaluate the optimal number of probes. Rademakers et al. [17] found that the number in radial as well in circumferential direction has significant influence on the determined distortion parameters. Hence, while comparing measurements to limits the circumstances under which these have been generated are very essential.

Besides the rakes providing data on the total pressure or swirl distribution in the AIP additional pressure taps can be installed to the inlet duct. As shown in figure 2 the measurement of the static wall pressure provides valuable information on the separation effects. When located at significant positions on the upper or lower surface of the inlet (only for symmetric inlet ducts) the separation onset can be determined very accurately. Especially for the validation of numerical simulations this kind of information can be very useful [1].

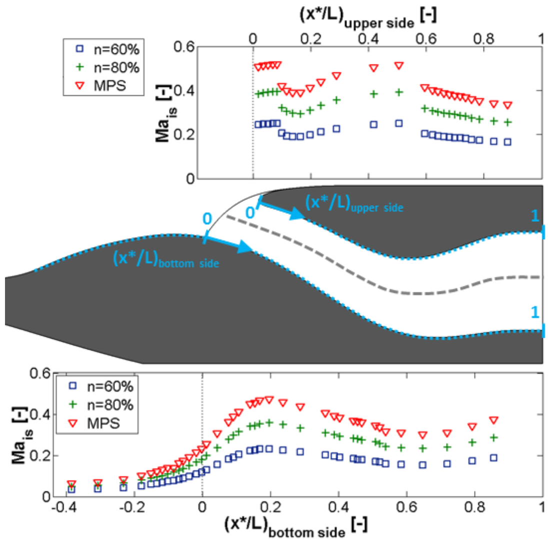

Figure 2: Results generated using pressure tap measurements

An example for the implementation of pressure as well as optical measurements into a compressor stage is described by Lieser et al. [12]. The instrumentation was installed within the different sections of the test rig to gather information of the propagation of distorted flows inside the stage. By rotation of the

distortion generator a high circumferential resolution could be achieved.
In addition the distortion itself, generated in front of the compressor, was measured by PIV. All the data are basis for the numerical work performed in parallel [14].

FURTHER OBJECTIVES

Like described previously there are important issues to be considered in order to adequately simulate or experimentally investigate inlet distortions and their influence on turbo compressors. Performing these investigations with common setups, however, is just revealing the phenomena partially. Hence, further boundary conditions have to be considered to achieve more representative results.

Flight envelope

As inlet flow phenomena and thus the impact on the engine vary by the boundary conditions at the inlet, especially Mach and Reynolds number effects are of particular importance. As modern unmanned platforms occasionally operate up to very high flight levels and in a wide range of speeds, these boundary condition variations should also be analyzed adequately. Therefore the influence of varying inlet conditions according to the flight envelope typical for subsonic aircraft operation have been investigated numerically by Brehm et al. [1].
The high number of the conducted simulations allows not only an overall prospective view how total pressure or swirl distortion parameters change within the envelope. It furthermore gives the opportunity to individually investigate the decoupled effects of varying Mach and Reynolds numbers on the aerodynamics within the duct and the developing distortions. Thereby both the distortion intensity as well as their spatial characteristics have been considered by using different distortion description parameters. Like exemplified in figure 3 the CDI parameter shows a nearly linear trend towards higher Reynolds numbers. As at low Reynolds numbers the flow separation starts at more upstream locations and is more intensive the impact on the distortion patterns at the AIP is expected to rise. That this effect is not of relevance for investigations with ducts of real dimensions in the desired flight envelope can be derived from the figure and was also proven by Berrier [18] who performed experiments with a boundary ingesting inlet.

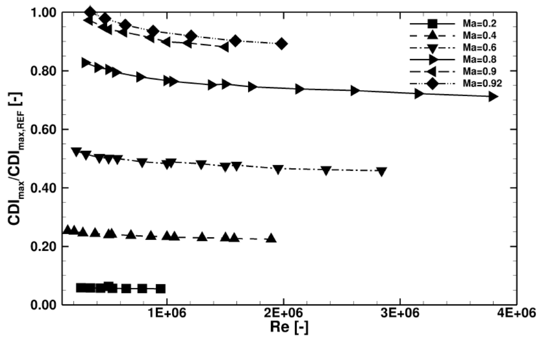

Figure 3: Variation of CDI parameter with Reynolds number

However, for experimental investigations Reynolds number scaling is of particular importance. For that reason Johnson [19] presents a methodology for scaling wind tunnel tests. Especially the frequency scaling for dynamic distortion has to be accounted in this context.
In contrast to the Reynolds number the flight Mach number is of importance for the performance of a serpentine inlet. As it can be seen in figure 3 the distortion at the AIP eminently increases with higher Mach numbers. In consequence additionally the pressure recovery is decreased as stated in [18].

Compressor interaction

Prior discussions mainly focused on the duct flows and in particular the separation phenomena which cause an inhomogeneous inflow to the fan stage. However, the interaction between the compressor and the inlet aerodynamics are of importance and have to be considered while assessing the propulsion system in total.
As the compressor rotor is designed for homogeneous pressure distribution at the inlet the disturbed flow conditions cause the stage to operate partially at off-design conditions. Due to the different operating points the flow field upstream of the compressor is affected by the rotor as well.
Looking at the interface between the inlet duct and the compressor in detail various effects can be distinguished. In nominal as well as disturbed operating conditions the shock waves of the transonic compressor blades propagate into the inlet if the interface of both is set at or near the leading edge.
Variations or even unsteady fluctuations of the static pressure field are generated by the com-

pressor when operated downstream of severe flow distortion. In consequence the compressor is often able to weaken the disturbances and therewith affecting the pressure field in the interface plane, which then is different to the one generated by the inlet alone.
Taking these effects into account especially numerical simulations or model based approaches have to consider coupling effects. Nevertheless, the accounting of blade shock waves will not improve the results in measurable quantities.
Not only the inlet is affected by the interaction but also the compressor. Spatially distorted intake flow or unsteady distortions in the intake result in differing operating conditions in circumferential direction and rotor stator interactions typically exhibit unsteady phenomena. Local significant changes in pressure ratio/mass flows and/or incidence can cause mechanical excitation of the blades, stall, or compressor surge [12].

ACTIVE AND PASSIVE FLOW CONTROL

In order to minimize secondary flow losses and inhomogeneous aerodynamic conditions at the engine inlet the avoidance of flow separation inside the serpentine duct is very promising. This can be achieved in an active as well as a passive way. Both configurations are able to provide a better flow field to the compressor.

- Passive countermeasures
  The flow conditions inside moderate bent inlet ducts, like installed in commercial aircraft, are well known and stabilization methods have also been applied several times. The most common method is the installation of small vortex generators to counter the merging boundary layer fluid and spread it evenly around the duct periphery. By using this passive method pressure recovery can be increased and flow distortions can be diminished. Commercial (e.g. Boeing 727) as well as military (e.g. F-111) aircraft are featuring this method.
  Furthermore, steady suction or blowing is utilized to affect the boundary layer and therefore the separation effects or even the secondary flows after they have established. Especially the latter is often subject of recent research. In combination with actuators blowing can also be used as active countermeasure.
- Active countermeasures
  The passive flow control methods can be very effective but are mostly optimized to certain operating conditions. Thus, at off-design operation the effectiveness decreases due to increasing losses. For applications with various operating conditions therefore active flow control can be an adequate solution, especially if it can be adjusted to the particular duct aerodynamics (engine mass flow rate, flight condition, etc.).
  In most investigated cases the inlet flow is affected by air blowing or boundary layer suction. That these methods can be very promising was demonstrated by Erbslöh and Crowther [20]. They compared conventional vortex generators with those realized by air jets. The experiments prove that pressure distortion at the engine interface could be reduced by 52% by the passive countermeasure and at least 34% by the air blowing. But in contrast to the conventional ones concerns like foreign object damage and icing are eliminated.
  A combination between suction and blowing can be achieved by using the ejector pump effect. Due to the already existing pressure gradients within the duct flow this can be an effective way. By utilizing higher pressure gradients and setting blowing aside even higher suction rates can be achieved. At the Institute of Jet Propulsion at the University of the German Federal Armed Force in Munich a combination of inlet suction and ejector exhaust nozzle is currently under investigation. One major advantage of the system is that not only the pressure gradient for the inlet suction but an additional air flow to cover the hot jet exhaust is provided. Nevertheless, the additional entrained mass flow can increase the thrust of the propulsion system.

SUMMARY AND CONCLUSION

Highly integrated propulsion systems claim more and more importance for future aerial platforms. While the next generation of commercial aircraft will feature conventional engine mounting configurations, the unmanned platforms and fighter aircraft today use bent

inlet duct and nozzle designs in order to achieve stealth capabilities. However, some design prognoses for future passenger aircraft also feature integrated propulsion systems.
During the last decades extensive research was performed on the field of flow conditions inside of contoured engine inlets. Improved numerical capacities nowadays allow detailed simulation of the aerodynamic phenomena within the ducts and furthermore the engines fan stage. Recent developments of measurement techniques are the key to the highly unsteady stall inception processes inside the compressor prone to inlet distortion. Evaluation of the established distortion parameters using the new possibilities determined a good basis to describe the inhomogeneous inlet flows. Certain shortcomings have been revealed if swirl distortion should be accounted for as well. Due to the recent inlet designs the swirl patterns differ from former bulk swirl configurations. Hence, the influence of those peak swirls on engine stability has to be evaluated carefully.
While performing future investigations on the topic some particular boundary conditions have to be taken into account to obtain more application-oriented results. Finally it can be stated that a lot of improvements have been made during the last decades but there is still a lot of potential for further efficiency enhancement.

REFERENCES

[1] Brehm, S., Kächele, T., Niehuis, R., "CFD Investigations on the Influence of varying Inflow Conditions on the Aerodynamics in an S-Shaped Inlet Duct", AIAA-2014-3595, Proceedings of the 50th AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit, July 28-30, 2014, Cleveland, OH, USA

[2] Wellborn, S.R., Reichert, B.A., Okiishi, T.H.,"An Experimental Investigation of the Flow in a Diffusing S-Duct", AIAA-92-3622, Proceedings of the 28th AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit, July 6-8, 1992, Nashville, TN, USA

[3] Sóbester, A., "Tradeoffs in Jet Inlet Design: A Historical Perspective" Journal of Aircraft Vol. 44, No. 3, May-June, 2007

[4] Cousins, W.T., "History, Philosophy, Physics, and Future Directions of Aircraft Propulsion System/Inlet Integration", GT2004-54210, Proceedings of ASME Turbo Expo 2004 Power for Land, Sea and Air, June 14-17, 2004, Vienna, Austria

[5] Muraru, A., "An Overview on the Concept of UAV Survivability", The 13th International Conference of Scientific Papers "SCIENTIFIC RESEARCH AND EDUCATION IN THE AIR FORCE" AFASES, May 26-28, 2011, Brasov, Romania

[6] Rademakers, R.P.M., Bindl, S., Niehuis, R., "Effects of Flow Distortions as They Occur in S-Duct Inlets on the Performance and Stability of a Jet Engine", GT2015-42062, Proceedings of ASME Turbo Expo 2015: Power for Land, Sea and Air, June 15-19, 2015, Montreal, Canada

[7] Zelenak, M., "An Uncertainty Analysis of Inlet Dynamic Flow Distortion using an Analog/Digital Hybrid Editing System", Final Report for 02/01/91-12/31/92, Aeronautical Systems Center, Wright Patterson AFB OH, March, 1993

[8] Ramachandra, S.M., Sudhakar, K., Perumal, P.V.K., Jayashima, P., "Air-Inlet Engine Matching Problems Encountered in a Jet Trainer Re-Engining Program", Journal of Aircraft, Vol. 19, No. 8, 1982, pp. 609-614

[9] Siller, U., Voß, C., "Automated Optimization of a Double S-Shaped Inlet for Minimum Loss and Reduced Sight onto the Engine Face", ISROMAC13-0032, The 13th International Symposium on Transport Phenomena and Dynamics of Rotating Machinery (ISROMAC-13), April 4-9, 2010, Honolulu, HI, US

[10] Reid, C., "The Response of Axial Compressors to Intake Flow Distortion", 69-GT-29, ASME Paper, 1969

[11] Seddon, J., Goldsmith, E.L., "Intake Aerodynamics", Collins, London 1985

[12] Lieser, J.A., Albrecht, P., Biela, C., "Planning of a compressor rig test with advanced inflow distortion simulation", Second Symposium "Simulation of Wing and Nacelle Stall", June 22-23, 2010, Braunschweig, Germany

[13] Longley, J.P., Greitzer, E.M., "Inlet Distortion Effects in Aircraft Propulsion System Integration", AGARD-LS-183, 1992

[14] Niehuis, R.; Lesser, A.; Probst, A.; Radespiel, R.; Schulze, S.; Kähler, C.J.; Spiering, F.; Kroll, N.; Warzek, F.; Schiffer, H.-P., "Simulation of Nacelle Stall and Engine Response", ISABE-2013-1402, Proceedings of the XXI. International Symposium on Air Breathing Engines (ISABE), September 9-13, 2013, Busan, Korea

[15] Hale, A., Chalk, J., Klepper, J., Kneile, K., "Turbine Engine Analysis Compressor Code: TEACC - Part II: Multi-Stage Compressors and Inlet Distortion", AIAA-99-3214, 17th AIAA Applied Aerodynamics Conference, June 28- July 1, 1999, Norfolk, VA, USA

[16] Society of Automotive Engineers (SAE), "Gas Turbine Engine Inlet Flow Distortion Guidelines", Aerospace Recommended Practice - ARP1420, Revision A, 1998

[17] Rademakers, R.P.M., Kächele, T., Bindl, S., Niehuis, R., "Approach for an Optimized Evaluation of Pressure and Swirl Distortion in S-Shaped Engine Inlet Configurations", AIAA-2014-3594, Proceedings of the 50th AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit, July 28-30, 2014, Cleveland, OH, USA

[18] Berrier, B.L., "Evaluation of Flush-Mounted, S-Duct Inlets with Large Amounts of Boundary Layer Ingestion", RTO Vehicle Propulsion Integration Symposium, October 6-9, 2003, Warsaw, Poland

[19] Johnson, R.H., "Inlet Distortion Scaling of Wind Tunnel Model Results", NASA CR-143840, 1976

[20] Erbslöh, S.D, Crowther, W.J., "Control of Boundary layer separation on a civil turbofan intake using air-jet vortex generators", CEAS/KATnet Conference on Key Aerodynamic Technologies, June 20-22, 2005, Bremen, Germany