NUMERICAL SIMULATION OF MIXED JET EXHAUST SYSTEM AND ITS VERIFICATION

J.A. Lieser, B. Deinert, C. Möller, F. Müller
Rolls-Royce Deutschland Ltd & Co KG
Dahlewitz, Germany

OVERVIEW

Forced mixing is used in BR700 and Tay long cowl engines for improved fuel consumption and reduction of noise. The advantages and design of mixed jet exhaust systems are described briefly. Numerical simulation is extensively used in the design process and numerical methods are described and results compared with experimental data. The aim of the present work was to extend computational fluid dynamics (CFD) from an analysis tool for certain engine conditions to a platform, which can deliver multidimensional nozzle characteristics over the whole range of the important parameters such as pressure ratios and temperature ratios. For this purpose methodologies of thrust determination were investigated and implemented in the design process. Examples for forced mixers and nozzle lip treatments are shown in the present paper.

1. INTRODUCTION

Rolls-Royce Deutschland BR700 and Tay families of aero engines are using mixed jet exhaust systems. This concept is typical for smaller engines installed at the rear of the fuselage on business jets and regional aircraft. The internal mixing and expanding of the hot core and cold bypass flow through a common nozzle increases thrust for small bypass ratio engines and significantly reduces noise. The axial length for flow mixing is limited by the overall length/weight of the engine. A forced mixer is used to intensify the mixing process. The goal for the mixer design is to accelerate the mixing at low additional losses and to avoid the occurrence of high frequency noise. By applying numerical methods in the design phase, the number of mixers to be tested experimentally was reduced from 5 to 1 during the last 10 years. The risk of high frequency noise generation has nearly vanished. Rolls-Royce Deutschland also investigates the use of nozzle lip treatment for internally mixed nozzles at present and future applications.

A German national research program on thrust improvement and noise reduction of forced mixer has lead to the complex scarfed mixer concept, which is used on the Tay 611-8C engine [1]. Basic studies on single and coaxial jets were performed in the European research projects JEAN and CoJeN [2,3]. The effect of nozzle lip treatments on mixed jets are investigated in the German national projects FREQUENZ/LEXMOS numerically and experimentally. Based on a detailed experimental database, gathered in the CoJeN project for coaxial jets of coplanar and short cowl configurations, computational fluid dynamics (CFD) studies were performed and tools validated. The methods were then applied in engine projects to mixed exhaust systems with forced mixers to calculate complete nozzle characteristics. The evaluation of experimental data for nozzle lip treatments and comparison with CFD are ongoing.

2. MIXED JET EXHAUST CONFIGURATIONS

A typical BR700 forced mixing configuration is shown in fig 1. For small engines of bypass ratios below 7 the mixing loss penalty is more than compensated by improved exhaust performance.

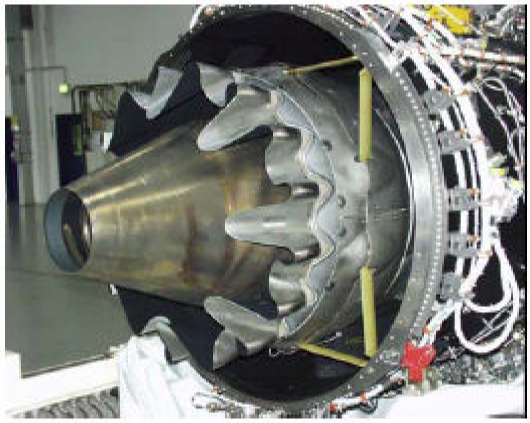


FIG 1. Forced mixer assembly

2.1. Engine Design and Modelling

The mixing process is part of the thermodynamic modelling of the engine and thus part of the cycle optimization. The mixing of the hot and cold streams upstream of the nozzle results in a thrust gain compared to separate expansion of the two jets [4, 5]. The two streams (core and bypass) are closely linked by the mixing process. While in separate jet configurations the fan and the core components can be matched by sizing the nozzles individually the coupling is much higher in the case of a mixed exhaust system. Equal static pressures at the mixer exit are assumed and the system is designed such, that the mixing losses in the shear layer upstream of the nozzle are minimized. There are different ways to model the exhaust system: A two stream model, were both streams virtually expanded separately and a three stream model were the streams are partially mixed and expanded as two unmixed and one fully mixed stream. The two stream model needs a velocity coefficient and effective nozzle exit flow areas for both streams. The three stream model needs common nozzle velocity and discharge coefficients (effective flow areas and geometric area) and in addition a mixing efficiency.

The velocity coefficient Cv of the nozzle is defined as the ratio of achieved exit velocity c_real to ideal exit velocity c_ideal or measured as gross thrust F_g divided by measured mass flow W and ideal exit velocity:

$$C_V = \frac{c_{real}}{c_{ideal}} = \frac{F_G}{W\ c_{ideal}} \quad (1)$$

The discharge coefficient is the ratio of measured mass flow to ideal mass flow:

$$C_D = \frac{W_{real}}{W_{ideal}} \quad (2)$$

The mixing efficiency η_{mix} is determined as the ratio of achieved thrust gain to ideal thrust gain. The thrust gain is the difference in thrust for a heated (index: hot) and an unheated (index: cold) core stream at identical nozzle pressure ratios. The achieved thrust gain is calculated from the difference of measured thrust with and without heated jet and the ideal thrust gain from the ideal velocity coefficients, where 100 % mixing is assumed for the hot case:

$$\eta_{mix} = \frac{\left(C_{v,hot} - C_{v,cold}\right)_{measured}}{\left(C_{v,hot} - C_{v,cold}\right)_{ideal}} \quad (3)$$

The efficiency is used in the three stream model to distribute mass flows to three streams:

$$\begin{aligned} W_{fan}^{\ *} &= (1-\eta_{mix})W_{fan} \\ W_{core}^{\ *} &= (1-\eta_{mix})W_{core} \\ W_{mixed}^{\ *} &= \eta_{mix}\left(W_{fan} + W_{core}\right) \end{aligned} \quad (4)$$

For the first one fan stream temperature is assumed, for the second one core temperature and for the third one ideal fully mixed temperature.

2.2. Forced Mixer

The goals of the mixer design are:

1. Maximize thermal mixing upstream of the nozzle
2. Minimize momentum loss
3. Provide engine matching as requested
4. Maximize mixing of hot stream (for noise reduction)

Mixing devices purely designed for noise reduction may compromise the specific fuel consumption. In [1,6] it was shown, that thrust gain and noise reduction can be achieved at the same time with a properly designed mixer. The scarfing and scalloping of forced mixers improve the mixing by introducing three dimensionality and vortex dynamics and reduce the turning and skin friction loss at the mixer itself. The so called complex scarfed mixer was patented by Rolls-Royce and first successfully applied in the TAY611-8C engine.

3. NOZZLE LIP TREATMENT

Nozzle lip treatments are well known as chevrons or nozzle serrations. There are physical and geometrical analogies with the forced mixer. In both cases the aim is to increase the mixing in shear layers. The forced mixer increases the area of the shear layer and generates longitudinal vortices. The nozzle lip treatment also increases the radial momentum transport, mainly by longitudinal vortices. The production of longitudinal vortices by conventional chevrons/serrations with an angle to the streamlines is most effective. Also combinations of mixer type nozzles and chevrons for mixers are possible like tabs at the nozzle and vortex generators on mixers [7].

The hot stream meets the outer shear layer downstream of the nozzle. The interaction is therefore influenced by the nozzle lip treatment. In this context the combination of mixed jets with nozzle treatment is of special interest.

4. NUMERICAL SIMULATION OF EXHAUST SYSTEMS

With the numerical simulation (CFD) one can study the influence of parameters like pressure and temperature ratios on nozzle performance individually. Due to the cost of a test with hot core stream the experiments used to focus on the measurement of points along an engine working line with all parameters varying from point to point. In addition test bed instabilities and measurement inaccuracies lead to data scatter. The investigation of small but systematic dependencies is therefore difficult. A statistical averaging requires a large number of measurements which is costly. Numerical simulations are reliable for determination of tendencies and the complete set of flow data enables the analysis and understanding of the physics.

4.1. Numerical Methods

The standard tool for the presented results is the commercial CFD software FLUENT. Meshing, spatial/time discretization models, turbulence models and different codes were investigated in the European programs JEAN (Jet Aerodynamics and Noise) [8] and CoJeN (Coaxial Jet Noise) [2] programs.

The Investigations showed that quadrilateral meshes give the most accurate results with a minimum of computational costs. Shear layers are best resolved by mesh lines aligned with the streamlines and normal gradients. Nevertheless it was not possible yet to generate a block structured hexahedral mesh for a highly scarfed mixer with an acceptable effort. Unstructured hexahedral meshes with hanging nodes were not tried yet for the forced mixer also they were promising in case of coaxial jets. The forced mixers were therefore meshed with CENTAUR hybrid prism/tetrahedral meshes (see FIG 2.).

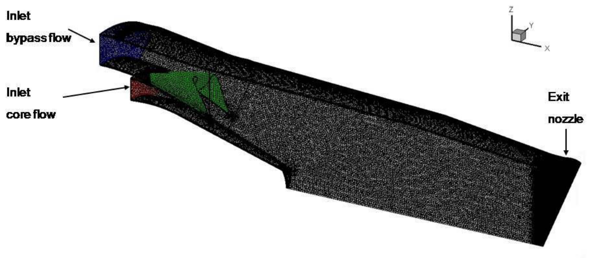


FIG 2. Computational domain and mesh for mixed exhaust system

Since most of the nozzle tests are static the numerical solver has to cope with zero Mach-number external flow. Therefore the FLUENT segregated solver was used in the present work. For the fluid properties ideal gas, Sutherland law and temperature dependent specific heat capacity was used. Laminar and turbulent heat transfer was linked to viscosity by the standard Prandtl numbers used in FLUENT.

The turbulence models available in Fluent version 6.2 were compared with respect to turbulent kinetic energy profiles and noise modelling [8]. The realizable k-ε and the Reynolds stress models showed the most promising results. Since the RSM is less robust with respect to numerical stability and difficult to get convergence the realizable k-ε is the standard model used for the forced mixer flows.

4.2. DETERMINATION OF THRUST

The CFD domain of a nozzle in most cases consists of the inner domain from nozzle entry to the nozzle exit plane and the far field. The interface in the nozzle exit plane can directly be used to calculate the thrust by integrating pressure and momentum. Since the forces in a closed control volume balance each other the thrust force is equivalent to all the forces acting on the remaining inner wall and entry boundaries. The same is valid for the outer control volume. There is not always a geometrical exit plane defined to calculate an integral and sometimes, as for serrated lips, is not easy to define. In addition it always has to be clearly defined which boundaries belong to inner (thrust) part and which belong to external forces (drag). The inner control volume for a mixed jet is shown in FIG 3.:

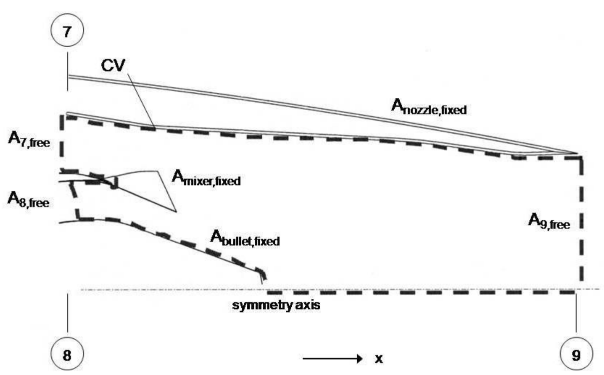


FIG 3: Inner control volume of a mixed jet

The integrals for calculating the forces from density ρ, velocity c and pressure p with the normal n for each surface A are shown in the following equation:

$$\int_{A_7} \rho \cdot \vec{c} \cdot (\vec{c} \cdot \vec{n}) \cdot dA_7 + \int_{A_8} \rho \cdot \vec{c} \cdot (\vec{c} \cdot \vec{n}) \cdot dA_8 + \int_{A_9} \rho \cdot \vec{c} \cdot (\vec{c} \cdot \vec{n}) \cdot dA_9$$

(5)

$$= -\int_{A_7} p \cdot \vec{n} \cdot dA_7 - \int_{A_8} p \cdot \vec{n} \cdot dA_8 - \int_{A_9} p \cdot \vec{n} \cdot dA_9 + \int_{A_{Bullet}} \vec{\sigma} \cdot dA_{Bullet} + \int_{A_{Nozzle}} \vec{\sigma} \cdot dA_{Nozzle} \int_{A_{Mixer}} \vec{\sigma} \cdot dA_{Mixer}$$

The integral over the boundaries of the outer flow field have shown to be less accurate. The integrals over the inner walls and inflow boundaries to the nozzle have shown to be the best way to calculate the thrust if an exit plane is not defined.

5. NOZZLE CHARACTERISTICS

The definition of the “ideal” reference is crucial for all coefficients. Normal practise is to use ideal expansion of the individual hot and cold stream to ambient conditions. For mixed streams the velocity coefficient is calculated by adding the product of measured mass flow times ideal exit velocity of both streams:

$$(6)\quad C_V = \frac{c_{real}}{c_{ideal}} = \frac{F_G}{\sum W\, c_{ideal}}$$

In our case a convergent divergent (condi) nozzle was investigated and the throat area is the reference area.

The “ideal” thrust gain is the increase in thrust, when the two streams are 100% mixed and then expanded through the nozzle without any mixing loss. For a constant pressure mixing it can be calculated analytically [5] and the result are shown in FIG 4. As can be seen from the analytical solution there is a dependency of mixing gain (F_mix/F_unmix) on cold stream nozzle pressure ratio (BPR), ratio of hot to cold temperature (TR) and fan to core pressure ratio (PR). The most significant dependence is on temperature.

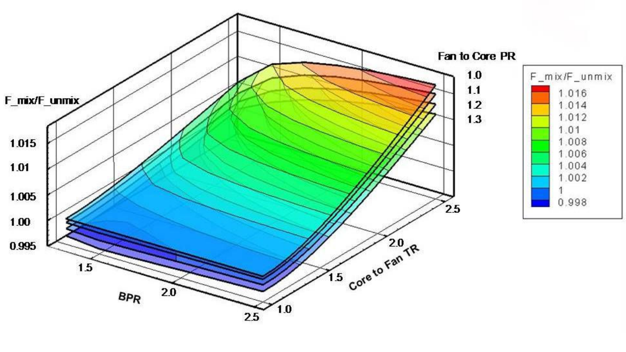


FIG 4. Ideal thrust gain from constant pressure mixing (from [5])

6. RESULTS

6.1. Coplanar Coaxial Nozzles

Most of the verification of the methods were done in the EU programs JEAN and CoJeN. The so called coplanar nozzle of CoJeN generates two parallel cylindrical shear layers originating at the same axial position. CFD studies were performed, where the outer nozzle end was moved downstream (similar to long cowl) and upstream (similar to a short cowl ,but without a plug). The axial position of interaction of the shear layers moved accordingly, but the flow fields and interaction looked very similar. The coplanar nozzle in CoJeN was investigated in the QINETIQ noise test facility (NTF) with particle image velocimetry (PIV) which gave very detailed data for comparison with CFD. Fig 5. shows a comparison of the velocity from CFD (upper) and experiment (lower). The potential core length in the CFD is longer (at around 10 diameters). FIG 6 shows a comparison of the turbulent kinetic energy. The maximum at the end of the potential core is similar but further downstream. It seems, that the interaction of the two shear layer happened further downstream in the CFD.

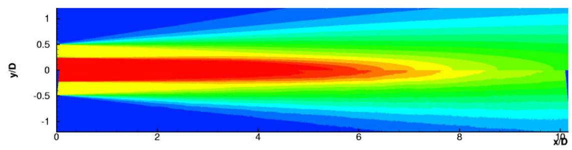


FIG 5. Comparison of calculated (upper) and measured (lower) velocity for a hot coaxial jet with coplanar nozzle exits

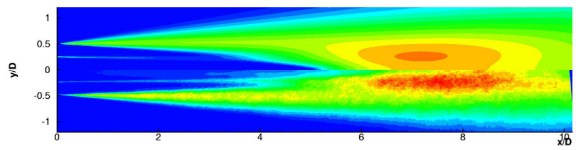


FIG 6. Comparison of calculated (upper) and measured (lower) turbulent kinetic energy for a hot coaxial jet with coplanar nozzle exits

6.2. Forced Mixer

Static thrust rig tests for a forced mixer configuration were performed at ASE FluiDyne facility. In advance of the systematic parameter study the test conditions were calculated by CFD and compared with the experimental results. In the presented CFD results the CFD domain ended at the nozzle exit and ambient static pressure exit boundary condition were set. A comparison of the velocity coefficient is shown in fig 7. The nozzle is a convergent divergent nozzle with a throat area slightly lower than the exit area. Close to choked conditions the flow gets transonic locally and the velocity coefficient gets worse around a pressure ratio of 2.0. The differences of CFD and experiment are within the test bed repeatability of 0.1%.

Fig 8. shows the comparison of discharge coefficients. The reference area is the throat area at all conditions. Therefore the coefficient rises for pressure ratio below choking. At a nozzle pressure ratio of around 2.1 the flow in the divergent part is expanding to ambient. Above 2.1 the nozzle is under expanded. Due to the boundary condition the difference of CFD to experiment rises with increasing nozzle pressure ratio.

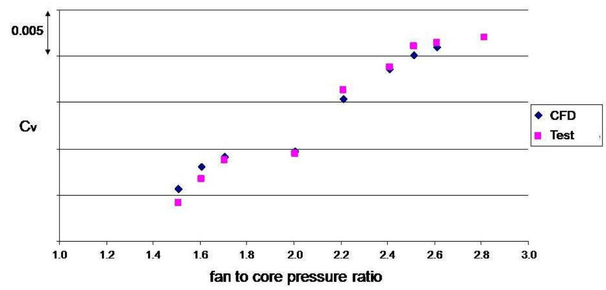


FIG 7. Comparison of velocity coefficients from experiment and CFD for hot mixed flow

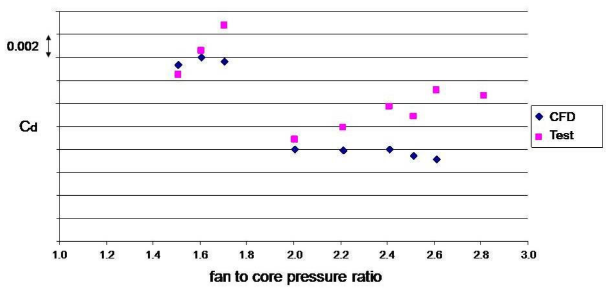


FIG 8. Comparison of discharge coefficient from experiment and CFD for hot mixed flow

Finally the mixed nozzle was calculated over a range of nozzle pressure ratios, cold to hot pressure split and temperature ratios and a mixing efficiency was calculated from that. The result is shown in fig 9.

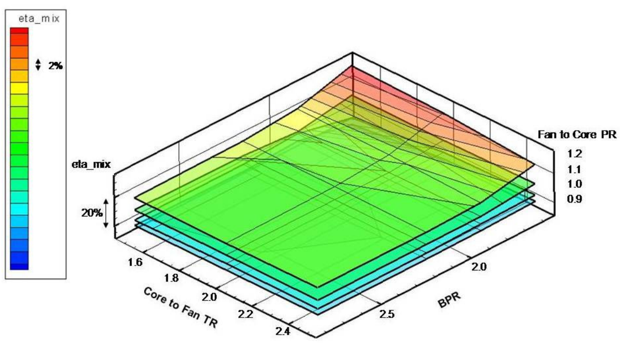


FIG 9. Mixing efficiency as calculated by CFD as a function of temperature ratio (TR), cold nozzle pressure ratio (FPR) and cold to hot pressure split (PS)

The efficiency is nearly independent of the temperature, since the ideal reference mixing gain (fig 4) already includes temperature. The mixing efficiency rises slightly with the nozzle pressure ratio. A systematic dependence on pressure split can be seen by increasing efficiency with pressure split.

6.3. Nozzle Serrations

In FREQUENZ nozzle lip treatment in combination with mixed exhaust systems were investigated to show the noise reduction capability of the lip treatment for mixed exhaust systems. The computation of the small scale flow treatment is very challenging. To capture the physics of the nozzle lip treatment a very good resolution of the flow around and downstream of the nozzle lip by CFD is necessary. It is also expected, that the nozzle characteristics become more difficult as, similar to the mixer, the discharge coefficient may become highly dependent on engine conditions.

Up to now only annular mixers were numerically investigated in combination with nozzle serrations. Two configurations of serrations were compared to a baseline nozzle without any lip treatment. 20 serrations are attached at the nozzle exit of the baseline. The shapes of the serrations investigated are identical except the inward bend angle (see table 1). SERRATED 1 is set at an angle of 0° and SERRATED 2 is set at an angle inwards relative to the jet axis.

TAB 1. Nozzle Geometries

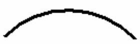
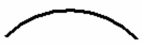
	BASELINE	SERRATED 1	SERRATED 2
Top View			
Front View			

Fig. 10 illustrates comparisons of the three configurations. Axial velocity contours are shown for four cross flow slices at downstream x/D values of 0.1, 0.3, 1 and 3. X is the axial distance starting from the serration trailing edge and D is the nozzle diameter. It can be seen that SERRATED 1 has only low impact on the shear layer compared to SERRATED 2. The results for SERRATED 2 show the distortion of the shear layer due to the vorticity generated. The so-called omega-structures can be observed. At x/D=3 the effect cannot be seen anymore and the three configurations show a similar circular shape.

A comparison of turbulent kinetic energy with and without serrations for the two types of serrations compared to the baseline configuration is shown in fig 11. The slices are along the plane of symmetry through the middle of a serration. Details for the distribution of the turbulent kinetic energy near the nozzle exit are shown on the left side of fig. 11. It can be seen that SERRATED 1 has no observable influence on the turbulent kinetic energy. In contrast, for SERRATED 2 the values of k increase near to the nozzle exit. This production of turbulence close behind the nozzle reduces the turbulence in the potential core as shown on the right side of fig. 11. A quantitative comparison of k is shown in fig. 12. Again the distribution shows that there is nearly no difference between the cases of BASELINE and SERRATED 1. Furthermore the reduction of the maximum of turbulent kinetic energy for SERRATED 2 can be noticed.

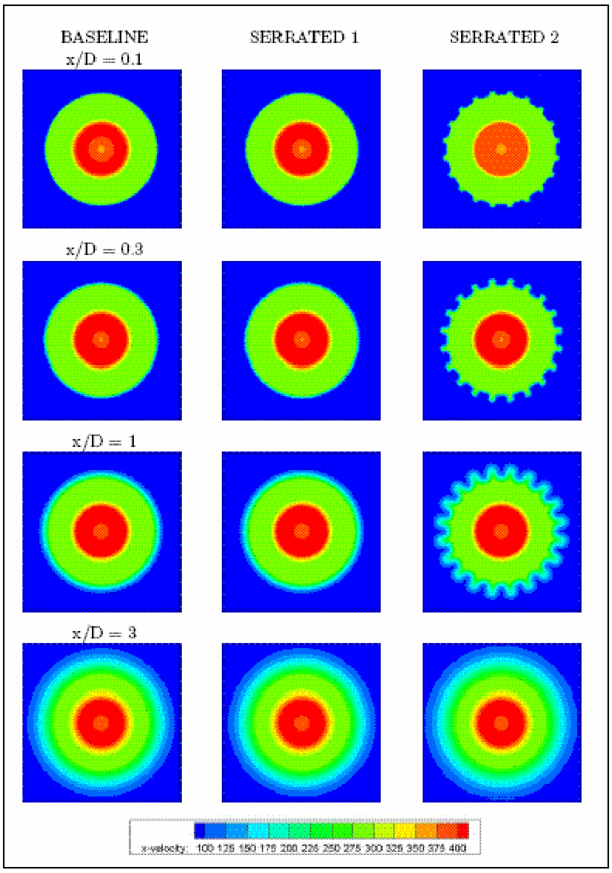


FIG 10. Axial velocity [m/s], baseline and serrated nozzles

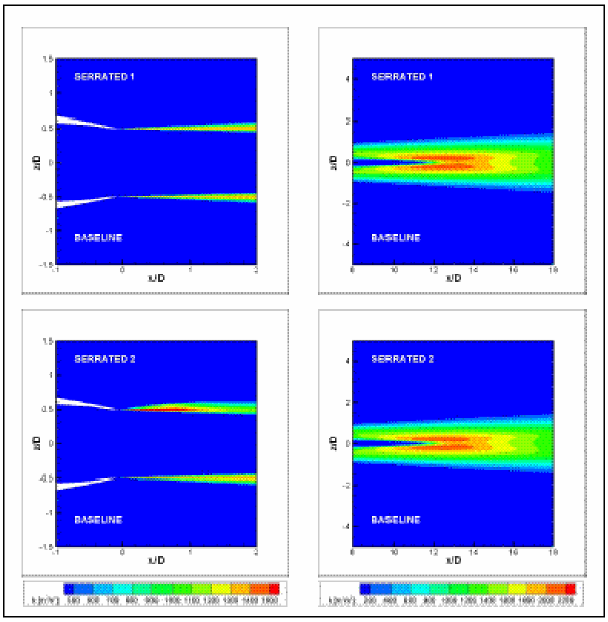


FIG 11. Turbulent kinetic energy contours along symmetry plane through serration centre

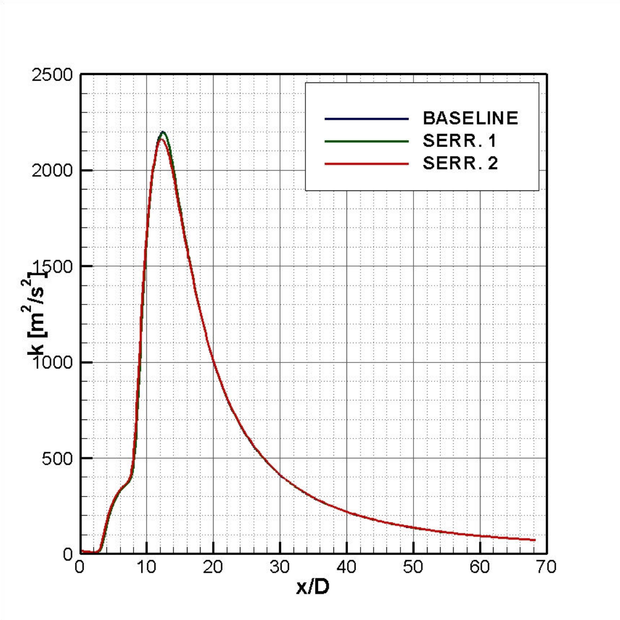


FIG 12. Distribution of centreline turbulent kinetic energy

Table 2 summarizes the changes in nozzle performance for the two types of serrated nozzles, including the discharge coefficient, thrust coefficient, changes in mass flow and thrust. For SERRATED 1 all parameters remain nearly unchanged. In case of SERRATED 2 the effective exit area is reduced due to the inward bend angle. Therefore mass flow and thrust decrease. Altogether the determined penalties are minimal.

	SERRATED 1	**SERRATED 2**
Δc_D [%]	-0.04	-1.78
Δc_V [%]	0.00	-0.01
ΔMass Flow [%]	-0.03	-1.66
ΔThrust [%]	-0.03	-1.71

TAB 2. Nozzle performances compared to baseline nozzle

7. CONCLUSION

By verification of the numerical methods with experimental results and use of CFD expertise from several research programs it is now possible to quantify nozzle performance by use of CFD. While the experiments suffer from statistical measurement uncertainties the CFD usually shows a deterministic effect. Therefore CFD was used to assess design iterations by calculated deltas and reduced the number of experiments and related costs. Also CFD is ideal for use in systematic studies of nozzle characteristics. Unlimited by test facility cost and availability it is easy to vary one parameter at a time and study the influence on performance. Analyse of the full set of data is possible to better understand the physics driving the shape of the aerodynamic characteristic of the nozzle system. Complete parameter variation studies show the importance or dependence of nozzle performance on certain parameters, which can vary from design to design. By use of numerical simulation aerodynamic characteristics of innovative nozzle systems can be defined with high confidence in an early stage of development.

8. REFERENCES

[1] M. Wegner, J.A. Lieser, Ch. Mundt: Lärm und Leistungsoptimierter Strahlmischer, DGLR-JT99-82, DGLR Jahrestagung, Berlin 1999
[2] N. Andersson, Lars-Erik Eriksson: Prediction of Flowfield and Acoustic Signature of a Coaxial Jet Using RANS-based Methods and Large-Eddy Simulation, 10th CEAS-ASS Workshop, September 2006, Trinity College, Dublin, Ireland
[3] J. Nogueira, M.Legrand, S. Nauri,Rodrigez, A. Lecuona: Analysis of the vortex street generated at the core-bypass lip of a jet engine nozzle, PivNet2 Final Workshop, DLR Göttingen, Germany, September 2006
[4] T.H. Frost, Practical Bypass Mixing Systems for Fan Jet Aero Engine, The Aeronautical Quarterly, 1966
[5] G.C. Oates: Aerothermodynamics of Gas Turbine and Rocket Propulsion, AIAA Education Series, 1988
[6] Ch. Mundt, J.A. Lieser: Performance Improvement of Propulsion Systems by Optimization of the Mixing Efficiency and Pressure Loss of Forced Mixers. 8th European Propulsion Forum, March 2001, Nottingham, 2001
[7] B. Deinert: Mischungsverstärker in konstant gekreuzten Scherschichten und im Abgasblütenmischer, Dissertation, TU-Berlin 2006
[8] J.A. Lieser, R. Spieweg: Strahllärmberechnung für gemischte Triebwerksstrahlen mit Hilfe akustischer Analogie. DGLR-Jahrestagung, München 2003
[9] W.A. Engblom, A. Kharavan, J. Bridges: Numerical Prediction of Chevron Nozzle Noise Reduction using WIND-MGBK Methodology, AIAA 2004-2979, NASA Glenn Research Center, 2004