COOLING STUDY IN THE EXHAUST DIFFUSER OF A REHEATED TURBOFAN

A. Matesanz, A. Viedma[*], A. Velázquez, M. Rodríguez[*]
SENER
PTM Tres Cantos, Madrid, Spain

[*]Also at the Universidad Politecnica de Madrid

ABSTRACT

The cooling circuit of the exhaust diffuser in a reheated turbofan engine is studied by using a theoretical approach. Results are compared with experimental data obtained from both calibration rig and engine test bed. It is found that the analytical method provides good agreement with the measured data. Then, it becomes a powerful, but easy to use, design tool.

LIST OF SYMBOLS

A	Cross-section area.
P	Pressure.
x	Spatial coordinate along the passage.
r	Radial coordinate.
M	Mach number.
T	Temperature.
Q	Net heat per unit mass added from external sources.
C_p	Specific heat at constant pressure.
f	Darcy's friction factor.
L	Perimeter.
D	Equivalent diameter. $\left(D = \dfrac{4\,A}{L} \right)$
γ	Specific heat ratio.

subscripts:

conv	convective
ext	external
imp	impingement
int	internal
rad	radiative

1. INTRODUCTION

This paper is aimed to analyse cooling aspects of the turbine exit guide vanes of a reheated turbofan engine. To do so, a design method is proposed that takes into account the relevant fluid features all along the cooling circuit and the available data in the open literature. The prediction method includes both theoretical analysis and semi-empirical correlations coupled to numerical simulation wherever further refinement is required. Cold flow and engine tests were also undertaken in order to check the accuracy of the analytical

predictions.

The geometry which has been analysed is a turbine exit case consisting of two parts: one static blade cascade used to minimise swirl and an annular diffuser aimed to reduce the flow Mach number to the reheat required conditions. The cooling system takes air from the by-pass duct and makes it flow through the inner passage of the blades. There, heat is evacuated from the solid walls to the cooling air. The trailing edge is locally cooled by capilar tubes. Nevertheless, most of cooling stream reaches the plenum chamber located inside the central body of the annular diffuser. From that plenum chamber air passes throughout a perforated plate and impinges right on the rear wall of the diffuser. This wall has to endure severe thermal conditions mainly due to radiative heat flux coming from the reheat. That is the reason why a very efficient cooling of impingement type is sought at that particular location. Finally, air cools down the lateral wall of the diffuser while flowing along its double skin and joins the main flow just downstream of the trailing edge. The fact that both parts of the turbine exit case share the same cooling circuit implies that analysis has to be performed in an integrated way. That must be so even though the dominant heat transfer mechanism is different in each region. A general overview of the geometry which has been studied is presented in Figure 1.

2. PHYSICAL MODEL

It has been pointed out in the Introduction that, for modeling purposes, the cooling circuit can be separated into four different parts. They are internal blade passage, plenum chamber, rear double wall of the diffuser and lateral double wall of the same diffuser.

The analysis has been made on a step by step basis so that the solution of each part of the circuit is used as an input to solve the next one. The fact that certain boundary conditions have to be met at both ends of the circuit means that an iterative procedure is needed to find out the solution of the problem. The boundary conditions which have to be met are both the total pressure and temperature at by-pass duct and the static pressure at the discharge holes of the cooling circuit.

The solution procedure starts by assuming a given mass flow going from the by-pass to the blade internal passage. Then, flow equations can be solved down
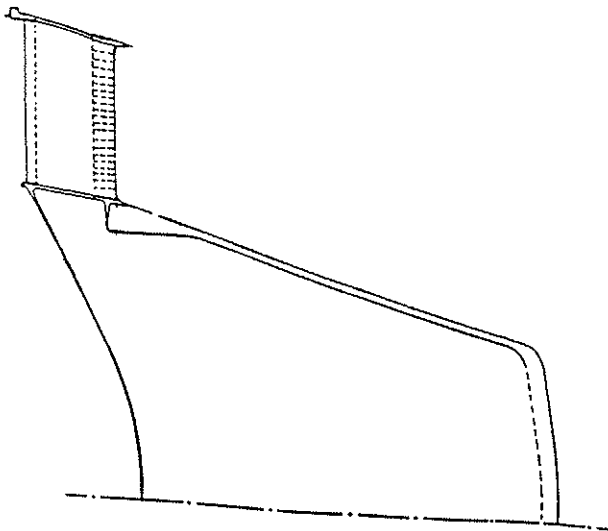

Figure 1. Exhaust diffuser geometry.

to the discharge holes. However, the discharge pressure will not be the required one so that an iterative method begins until that boundary condition is fulfilled within a given accuracy. A predictor-corrector method is used to speed up the scheme convergence.

2.1 Internal blade passage

To capture air from the by-pass in a efficient fashion, the cooling system has an intake whose shape has been designed to minimise pressure losses. Nevertheless, there is a finite drop in pressure because of the intake geometry which suddenly compresses and expands the air flowing through [1].

The flow is assumed to be one-dimensional inside the internal blade passage because of the fact that its length is large compared with its equivalent diameter. The equations governing the problem are as follows [2]:

$$\frac{dP}{P} = -\frac{\gamma M^2}{1-M^2}\cdot\frac{dQ}{C_p T} - \frac{(1+(\gamma-1)M^2)\gamma M^2}{1-M^2}\cdot\frac{f\,dx}{2D}$$

$$\frac{dT}{T} = \frac{1-\gamma M^2}{1-M^2}\cdot\frac{dQ}{C_p T} - \frac{\gamma(\gamma-1)M^4}{1-M^2}\cdot\frac{f\,dx}{2D}$$

$$\frac{dM^2}{M^2} = \frac{1+\gamma M^2}{1-M^2}\cdot\frac{dQ}{C_p T} + \frac{2\gamma M^2+\gamma(\gamma-1)M^2}{1-M^2}\cdot\frac{f\,dx}{2D}$$

The mass flow along the internal passage of the blade changes because of air escaping throughout the cooling capilar tubes. However, each bleeding is much smaller than the mainstream mass flow (of the order of 1%). Then equations (1-3) can be solved with constant mass flow at duct regions located between two capilars. At the junction points another iterative procedure starts in order to find out the right bleeding which leads to the appropriate static pressure at exit. The corresponding pressures drops are self-consistently calculated. A NASTRAN code was used to calculate the metal temperature of the blade. Flow parameters were given as an input to the numerical scheme. Q and f were calculated according to references [3-6].

2.2 Plenum chamber

The plenum chamber has the biggest volume and the largest cross-section area of the cooling circuit. Then, due to the fact that velocity in the plenum chamber is much smaller than in the remaining parts of the circuit, spatial variations of pressure are negligible compared with total pressure drop. Taking into account that simplification, the mass and energy conservation equations are used to compute the temperature in the chamber.

2.3 Rear double wall of the diffuser

The dominant feature in this part of the cooling circuit is the combination of cross-flow and impingement cooling. However, the fact that the rear wall has radial geometry somehow simplifies the mathematical model. Equations (1-3) can be used again with the appropriate change to polar coordinates:
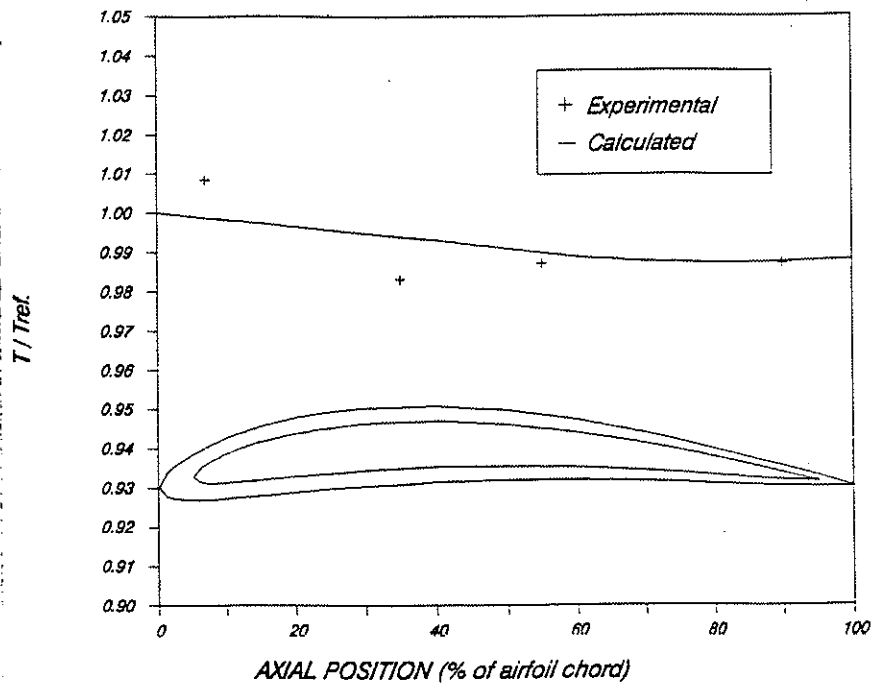
Figure 4. Comparison between predicted and experimental values of blade
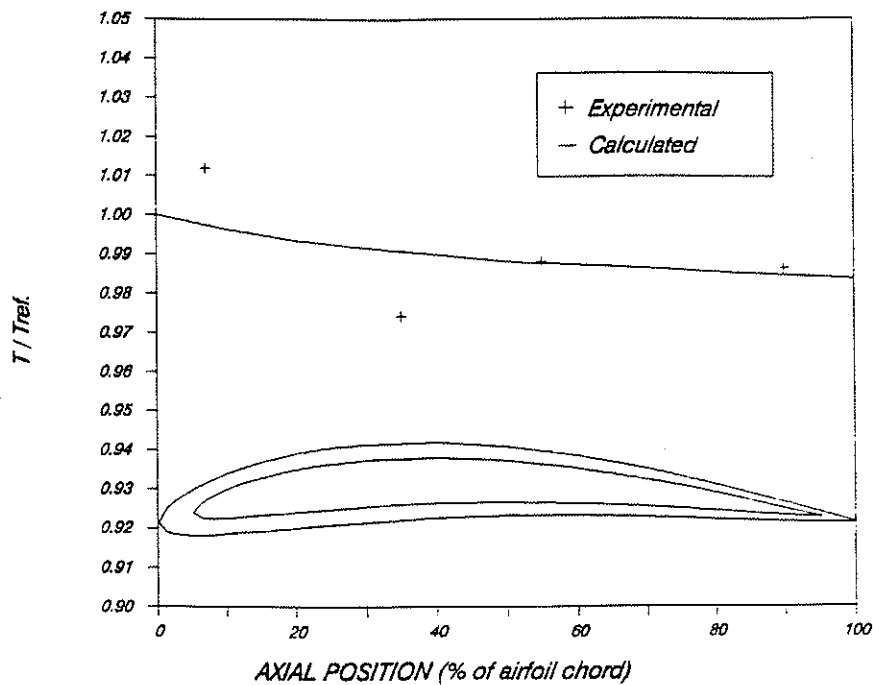temperature. (Calculated data were obtained by NASTRAN code)

Figure 5. Comparison between predicted and experimental values of blade
temperature. (Calculated data were obtained by NASTRAN code)

Figure (4) and (5) show the comparison between the predicted metal surface
temperatures and the experimental ones for two different test points. Diffuser
temperatures at three locations during a running line are presented in Figure
(6), (7) and (8). Finally, Figure (9) shows an overview of the accuracy of the
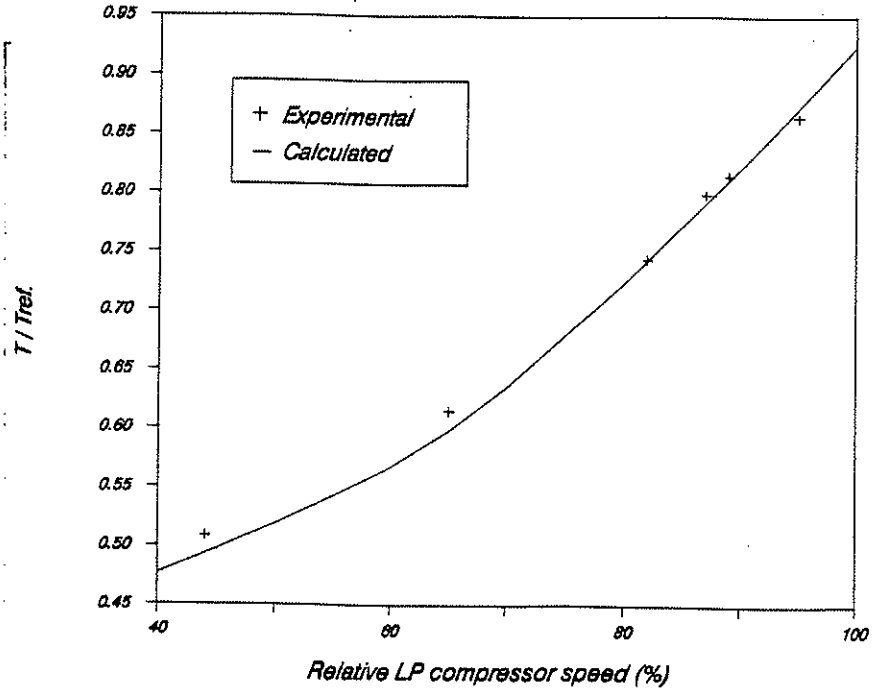theoretical model.

Figure 8. Comparison between predicted and experimental values
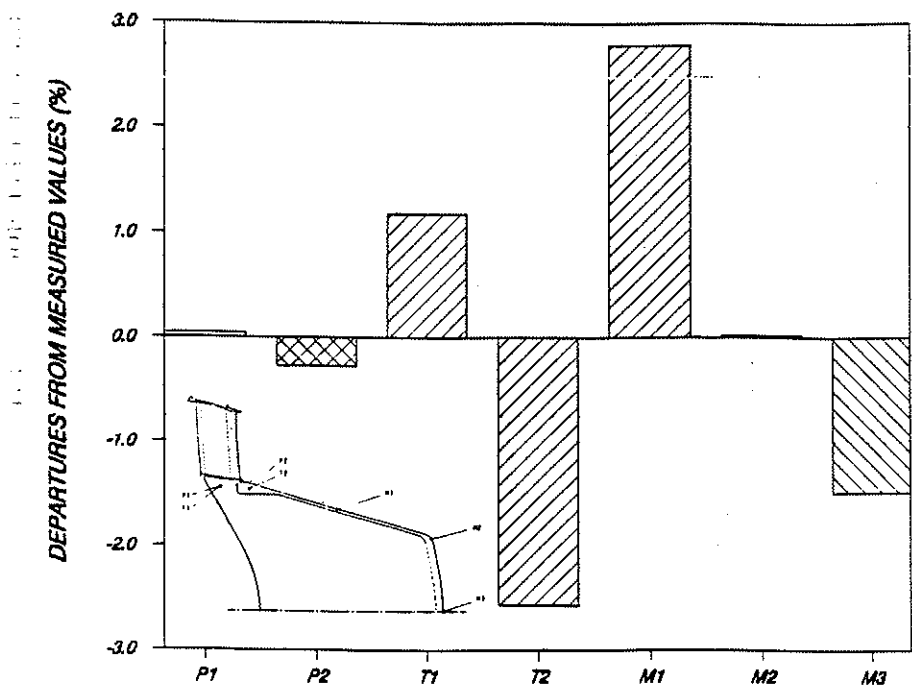of temperature at an intermediate section of the lateral wall.

Figure 9. Results of engine validation test. P, T and M are pressure,
air temperature and metal temperature respectively.

4. CONCLUSIONS

The previous study shows that it is possible to design the cooling circuit of a turbine exit case in a reheated engine by using available data from the open literature and the adequate analysis for the different flow regimens.

The results show a good agreement between the theoretically predicted values and the ones obtained both in rig calibration and engine test bed. Relevant engineering design parameters such as metal temperatures, pressures and mass flow were predicted within a 5% accuracy. Then, it is possible to explore a broad range of flight envelope points in a rather straightforward way by using this simple design tool. Further improvements on the modeling of reheat radiative properties would be needed if greater accuracy were required.

The experience has shown that the mean metal temperature values calculated from our method are accurate enough to design walls where heat conduction could be neglected. For blade-like geometries, this model provides the needed boundary conditions to compute numerically the thermal balance of the walls.

5. REFERENCES

[1] Adkins and Gueroui. 31st International Gas Turbine Conference and Exhibit, 86-GT-149.

[2] A.H. Shapiro. The Dynamics and Thermodynamics of Compressible Fluid Flow. Ronald Press Co, New York, 1953, Vol. I.

[3] S.E. Haaland. J. Fluid Eng., Vol 105, pp 89-90, 1983.

[4] Liang and Fairbanks. Transfer investigation of laminated turbine aircraft", ASME gas Turbine Heat Transfer 1978.

[5] M.N. Ozisik. Heat Transfer, a Basic Approach, Mc Graw-Hill.

[6] Webb, Eckert and Goldstein. Int J. Heat and Mass Transfer, vol 14, pp 601-617, 1971.

[7] Kolodzie and Van Winkle. Discharge coefficient through perforated plates. General Electric, Fluid Flow Division.

[8] Florschuetz and Isoda. J. Eng. for power, vol 105, pp 296-304, 1983.

[9] Hay and Spencer. International Gas Turbine and Aeroengine Congress and Exposition 91-GT-269.

[10] Kercher and Tabakoff. ASME J. of Eng. for Power, Jan 1970, pp 7391-92.

[11] Obot and Trabold. Transactions of the ASME, vol. 109, Nov. 1987.

[12] Dabagh et al. Gas Turbine and Aeroengine Congress and Exposition, 89-GT-188.

[13] Husain and Andrews. Gas Turbine and Aeroengine Congress and Exposition, 90-GT-285.

[14] Florschuetz and Su. Trans. ASME, vol. 109, Feb. 1987.

[15] A.H. Lefebvre. Gas Turbine Combustion. McGrawn-Hill, 1983.

[16] A.J. Chapman. Heat Transfer, Mac Millan, New York 1984.