THE AMERICAN SOCIETY OF MECHANICAL ENGINEERS
Three Park Avenue, New York, N.Y. 10016-5990

99-GT-237

ROLLS ROYCE RB211-DLE COMBUSTOR DIFFUSER DESIGN OPTIMISATION

BREAK

Tom Schweiger
Rolls-Royce Industrial & Marine
Gas Turbines Ltd.
Ansty, Coventry, CV7 9JR, UK
Tel: +44 (0)1203 623381
Fax: +44 (0)1203 623520
e-mail: tom.t.schweiger@rolls-royce.btx400.co.uk

Richard M Underhill
Duncan W Livingston
Frazer-Nash Consultancy Ltd.
1 Trinity Street, Bristol, BS1 5TE, UK
Tel: +44 (0)117 922 6242
Fax: +44 (0)117 922 6524
e-mail: r.underhill@fncb.co.uk

ABSTRACT

This paper presents a technique for optimising the performance of a diffuser in an industrial gas turbine using validated CFD modelling. The combustor module of the Rolls Royce RB211-DLE industrial engine was modelled from diffuser inlet to combustor inlet, using a hybrid meshing procedure. A CFD model of the current RB211-DLE diffuser and casing was validated against perspex single sector rig data, including pressure probe measurements, oil dot flow tests and a sensitivity analysis. A three-dimensional design process was then undertaken to determine how the shape of the diffuser affects the loss through the system, and hence which type of diffuser would provide the best opportunity for maximising the engine performance. The best two general diffuser designs were optimised using an iterative two-dimensional design process. The performance of these optimised designs was then confirmed by full three-dimensional modelling. This work suggests that a significant improvement in sfc (based on a constant turbine temperature) would be achieved if the optimum diffuser design is installed into the RB211-DLE engine.

INTRODUCTION

The Rolls-Royce RB211 industrial gas turbine has been in service for many years. Recognising the need to provide a low emissions engine, the RB211 was the first lightweight gas turbine to incorporate a new Dry Low Emissions (DLE) combustor. To enable the shaft lengths to be kept constant, the main difference between the RB211 and DLE version occurs in the combustor module (Fig. 1), where the high pressure compressor (HPC) flow is diffused and turned radially outwards before entering the combustor. The existing design includes a rectilinear diffuser, derived from 1D calculations and model testing (Willis, 1983) to achieve the required flow quality for good emissions.

In the constant drive for higher performance industrial gas turbines, better efficiency has generally been achieved by optimising turbine and compressor blade designs. However, the combined turning and diffusion within the combustor module, together with the blockage caused by the discharge nozzle, represents a region of high loss within all industrial gas turbines. It was recognised that the current design had relatively high pressure losses, and so significant improvements in engine performance could be achieved by optimising the shape of the diffuser within the combustor module. Since the diffuser performance and flow profile is dependant on the downstream blockage, normal throughflow calculations and turbine design methods are unsuitable, so the diffuser design has historically been based on standard empirical data (ESDU, 1975) combined with experimental tests.

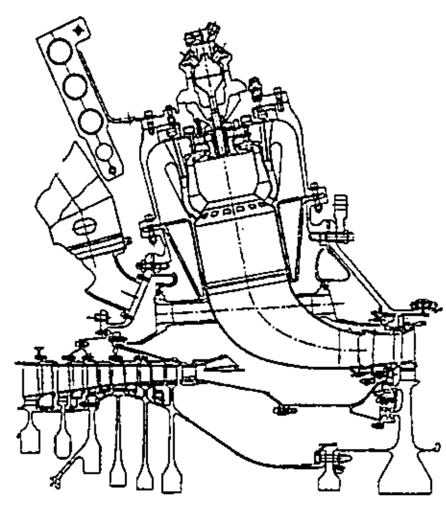


Fig. 1 - Drawing of the RB211-DLE combustor module

To improve the Rolls-Royce RB211-DLE diffuser, a new approach was adopted using the commercial CFD code Fluent/UNS. The objective was to optimise the diffuser design, and so minimise the pressure loss from diffuser inlet to combustor inlet. An additional secondary requirement was to improve the flow distribution to the combustor, and so reduce emissions further. The technique that was applied represents a multi-stage optimisation process that can be applied to any industrial gas turbine. This method contains four distinct phases, which are summarised in the flow diagram given in Fig. 2.

These four phases, applied to the Rolls-Royce RB211-DLE diffuser, demonstrate that significant improvements in diffuser performance are possible using this design method.

MODELLING DETAILS

The HPC air enters the RB211-DLE combustor module through an annular diffuser, turns radially outwards and separates into nine combustor cans. The CFD model is based on a perspex single sector test rig, which represents one combustor can (40° section) with side

Presented at the International Gas Turbine & Aeroengine Congress & Exhibition
Indianapolis, Indiana — June 7–June 10, 1999
This paper has been accepted for publication in the Transactions of the ASME
Discussion of it will be accepted at ASME Headquarters until September 30, 1999

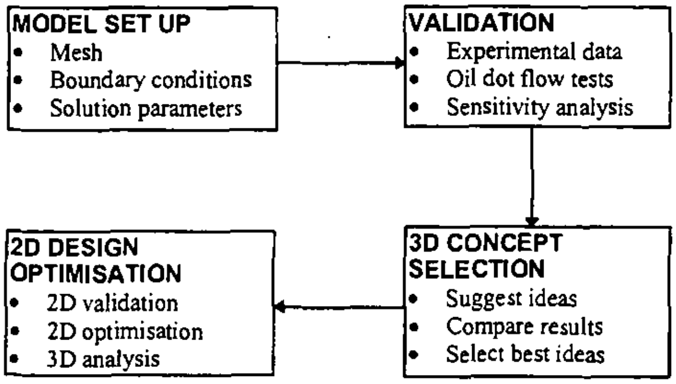


Fig. 2 - Flow diagram of design process

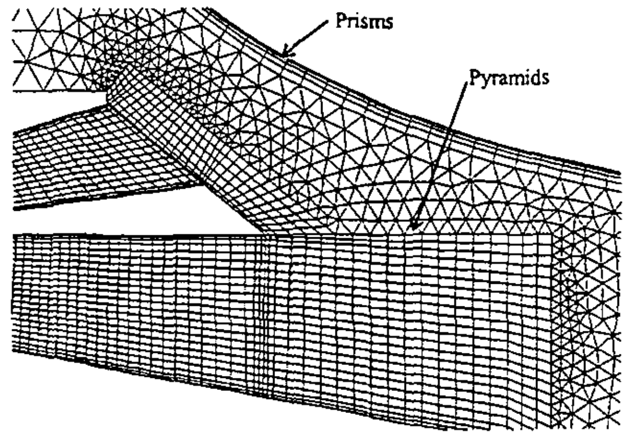


Fig. 4 - Mesh on symmetry plane of RB211-DLE model

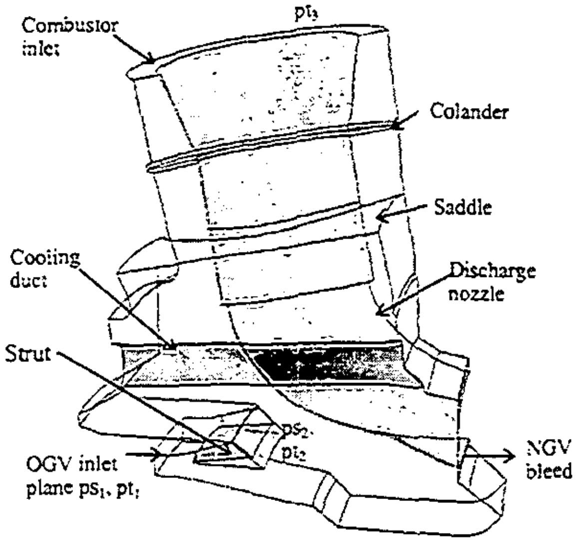


Fig. 3 - Geometry of the RB211-DLE CFD model

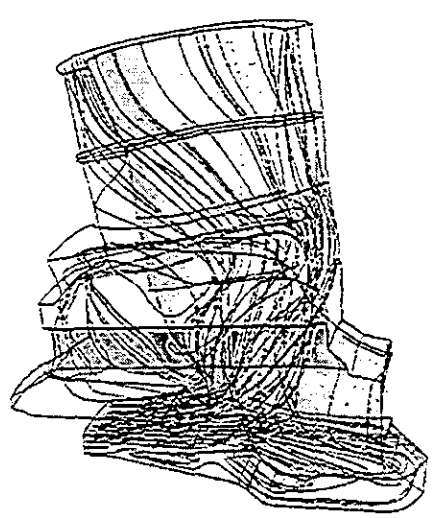


Fig. 5a - Pathlines released from the diffuser inlet

walls situated midway between combustor cans.
The CFD model covers a 20° section and assumes that asymmetric flow effects within each combustor are negligible.

The geometry used to perform the analysis is a 3D model of the current RB211-DLE split diffuser and discharge nozzle. The geometry was transferred to the GEOMESH suite of programs from CADDS in IGES format. The casing was simplified to improve the mesh and remove unwanted areas. The CFD model (Fig. 3) extends from the compressor outlet guide vane (OGV) inlet plane to the combustor inlet plane, incorporating all components affecting the combustor external flows.

The model was created from three separately meshed sections (diffuser, casing and combustor), to allow different diffuser designs to be quickly analysed. This was achieved using a hybrid meshing procedure (Fig. 4) with conformal interfaces between sections joined together using layers of pyramids.

The diffuser section contains a high density hexahedral mesh (100,000 cells), while the casing region uses tetrahedral cells, with four layers of prisms grown from the surface of the discharge nozzle and cooling duct (160,000 cells). Finally, the annulus around the combustor contains a hexahedral mesh (40,000 cells). The discharge nozzle prism heights and bunching within the diffuser were adjusted to obtain the required y+ values for the non-equilibrium wall function.

The boundary conditions applied to the model simulate the flow through the perspex single sector test rig. The inlet velocity profile, specified at the OGV inlet plane of the CFD model, was based on a Kiel probe traverse measurement from the perspex single sector rig using a parallel section substituted for the diffuser. The corresponding turbulence kinetic energy and dissipation rate profiles were calculated using a separate CFD model, where the velocity profile was fixed and the turbulence profiles allowed to develop. This enabled the velocity profile to be kept constant, while the turbulence parameters adjusted themselves to suit the velocity profile input.

The bleed to cool the turbine nozzle guide vanes (NGV) was represented by a velocity exit condition, i.e. correct mass flow rate, situated below the discharge nozzle. The CFD model outlet (combustor inlet) was defined as an outflow condition, rather than over-constrain the flow using a constant static pressure boundary. A perforated annulus plate (colander) is located within the combustor annulus, to re-distribute the flow entering the combustor. This was represented as a porous medium in the CFD model, with the pressure drop in the flow direction calculated from an empirical relationship (Miller, 1990).

The analysis was performed using a second order steady state viscous flow analysis with the standard k-ε turbulence model and non-equilibrium wall function. Heat transfer effects were included and density was calculated by an ideal gas law. The material properties, operating pressure and temperature were based on the atmospheric conditions within the perspex single sector rig.

GENERAL FLOW PROFILE

Before validating the model or suggesting alternative designs, it was necessary to understand the main flow features (Figs. 5a to 5c) predicted by the simulation. The results demonstrate that some of the

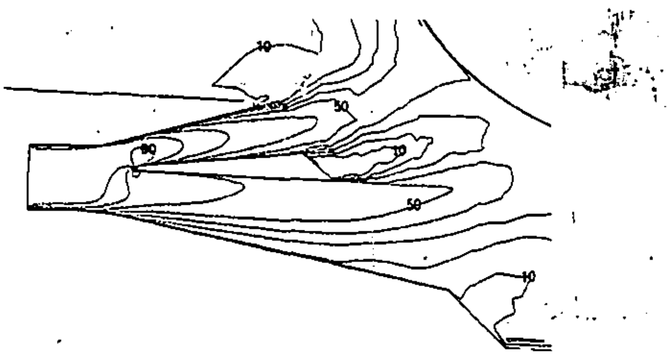


Fig. 5b - Velocity contours (increment 10 m/s) along Cut-2 of in-service diffuser

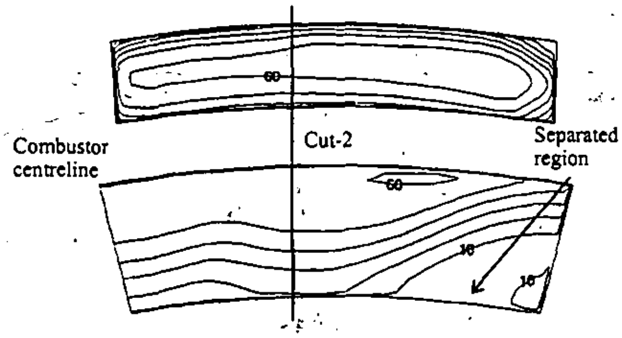


Fig. 5c - Velocity contours (increment 10m/s) at diffuser exit for the RB211-DLE CFD model

air exiting the diffuser impacts on the surface of the discharge nozzle and is driven underneath, creating a region of recirculating flow beneath the discharge nozzle. This recirculating region interrupts the flow exiting the inner channel of the diffuser midway between the combustors, causing the flow to separate off the inner wall.

The recirculating flow, combined with the outer channel flow, is then forced up through the gap between the combustors and re-accelerates due to the presence of the cooling duct. As the flow passes the cooling duct, most of the air flows into the turbine side of the combustor annulus. The remainder impacts on the saddle and then recirculates in the calm region above the diffuser before entering the compressor side of the combustor annulus. The colander plate enables some re-distribution to occur leaving a fairly uniform flow profile at outlet, which is slightly biased towards the turbine side of the combustor annulus.

VALIDATION

The datum model was validated against experimental data, by comparing diffuser exit profiles and pressure loss measurements, together with near-wall flow predictions using oil dot flow tests. Further confidence in the CFD predictions was achieved by conducting a sensitivity analysis of the main parameters within the model.

Pressure Measurements and Losses

The rig tests were conducted on a full scale perspex single sector rig, using a well instrumented diffuser (Fig. 6). The static and total

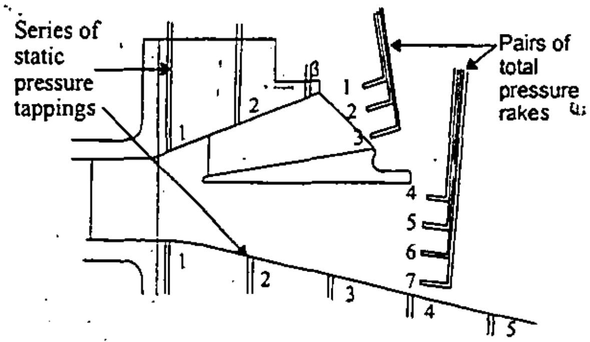
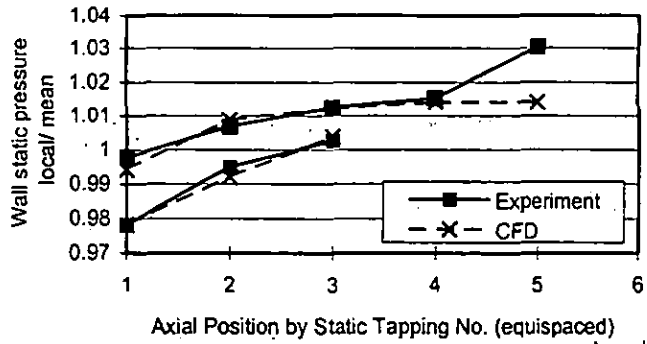


Fig. 6 - Outline of Diffuser and location of pressure probes



Fig, 7a - Static pressure profile on diffuser walls

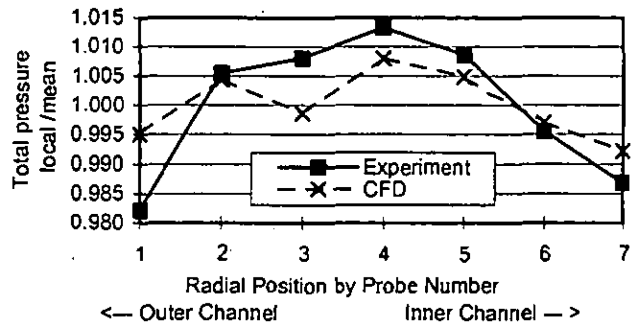


Figure 7b - Total pressure profile at the diffuser exit

pressure measurements were taken at three circumferential positions. A dummy diffuser with parallel walls was used to estimate the total pressure and velocity profiles at inlet to the diffuser.

Since the aim of the CFD analysis is to compare diffuser performance, it is important to predict the flow through the diffuser as accurately as possible. The validation of the diffuser predictions was achieved using the static pressure profile along the inner and outer walls of the diffuser (Fig. 7a), combined with the total pressure profile at diffuser exit (Fig. 7b).

The total pressure at each of the probe positions was extracted from the CFD solution, and the results are plotted non-dimensionally i.e. local/mean.

local = average of the three circumferential readings.
mean = average of all pressure probes.

The CFD mass flow predictions agree with the rig measurements, although the flow profile in the outer channel should be more biased

Nomenclature

AR	=	area ratio
C_{pr}	=	static pressure recovery coefficient = $(ps_2 - ps_1)/q_1$
K	=	total pressure loss coefficient = $(pt_1 - pt_3)/q_1$
K_{diff}	=	diffuser loss = $(pt_1 - pt_2)/q_1$
K_{dump}	=	dump loss = $(pt_2 - pt_3)/q_2$
ps	=	static pressure
pt	=	total pressure
q	=	dynamic pressure
θ	=	channel exit angle

Subscripts

1	=	inlet plane (OGV inlet)
2	=	diffuser exit plane
3	=	outlet plane (combustor inlet)
in	=	inner channel
out	=	outer channel

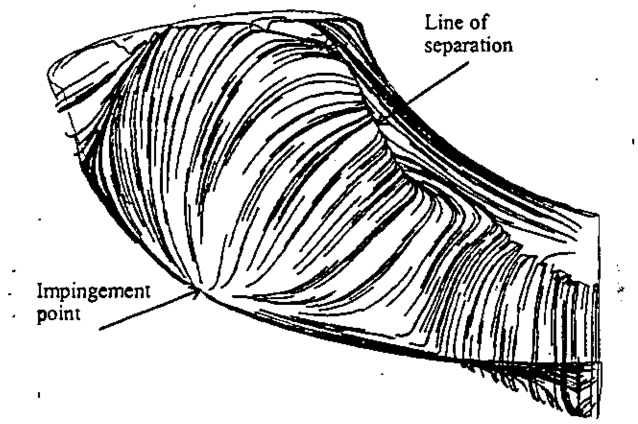


Fig. 8a - Predicted near-wall velocities on discharge nozzle

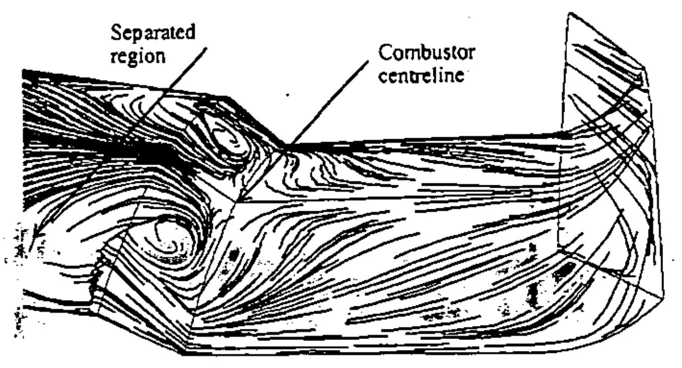


Fig. 8c - Predicted near-wall velocities on CCIC surface

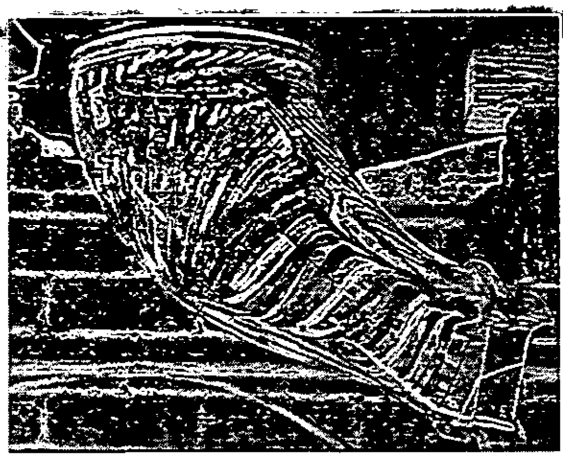


Fig. 8b - Oil dot flows on the discharge nozzle

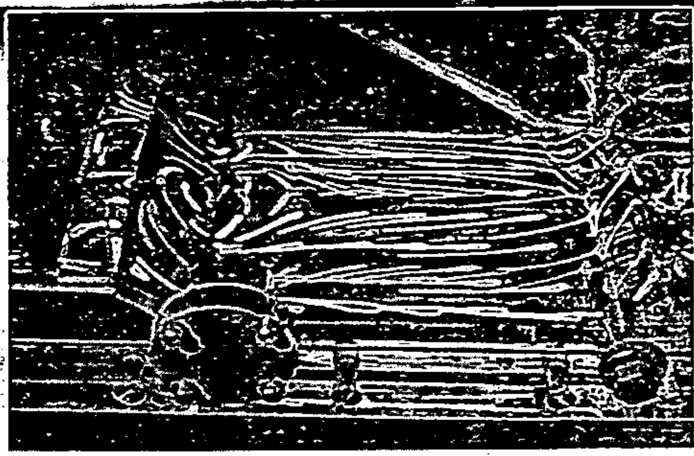


Fig. 8d - Oil dot flows on the CCIC surface

Table 1 - Pressure loss predictions for datum CFD model

Results expressed as % change from perspex rig probe data	Datum CFD model	
	Probe data	Mass averaged data
Diffuser loss, K_{diff}(%)	-3.4	-35.2
Diffuser recovery, C_{pr} (%)	+14.0	+19.8
Dump loss, K_{dump} (%)	0.0	+3.6
Overall loss, K (%)	-11.7	-11.7

towards the splitter. Since the flow is moving radially into the outer channel, it separates off the point of the splitter, which means that the inlet profile does not accurately simulate the perspex single sector rig. Since the inlet profile was measured using a parallel section, the inlet profile will change when the RB211-DLE diffuser is inserted.

The discrepancies in the inner channel static and total pressure profiles occur due to the optimistic separation predictions by the CFD model. This may be due to the limitations of the standard k-ε turbulence model or the accuracy of the inlet profile.

These discrepancies will be hard to eliminate, since no further data is available to improve the inlet velocity and turbulence profile. Although all turbulence models have limitations, the CFD analysis was set up to produce the best possible results from the applied model.

The pressure losses in the perspex single sector rig were calculated from the averaged measured probe data, and so only represent an estimate of the actual loss within the diffuser. The CFD predictions based on the average of the rig measurement positions, together with the more accurate mass averaged (ps_1, pt_1, ps_2, pt_2, pt_3) loss predictions, are shown in Table 1. The difference between the CFD predictions of the pressure at the probe positions and the mass averaged data indicates the expected discrepancy between the measured and real diffuser loss for the perspex single sector rig.

The probe data results confirm that the CFD model is accurately predicting the diffuser and dump losses, while the main difference in diffuser recovery and hence overall loss is probably caused by the problem highlighted previously i.e. separation prediction and inlet boundary profile.

Surface Flows

Oil dot flow tests were conducted on the external surface of the discharge nozzle and combustion chamber internal casing (CCIC) surface of the perspex single sector rig at normal operating conditions. These surface flow profiles were compared against the equivalent CFD near-wall flow predictions to provide additional validation for the CFD model. The oil dot flow photographs suggest that some asymmetry exists in the flow around the discharge nozzle, which could have been due to a leak in the system.

Figures 8a to 8d confirm that the main flow features present in the oil dot flow photographs are predicted by the datum CFD model. The flow impinges at a single point on the discharge nozzle centreline (Figs. 8a and 8b), and is then forced beneath the discharge nozzle and recirculates (Figs. 8c and 8d), causing the flow to separate off the inner diffuser wall midway between combustors.

Sensitivity Analysis

As part of any CFD analysis, a study should be undertaken to determine whether the results are sensitive to model flow parameters. Since the model is quite large, the number of cells could not be increased dramatically, so the mesh was checked and locally adapted to confirm that the correct mesh density, near-wall bunching and cell size distribution were appropriate.

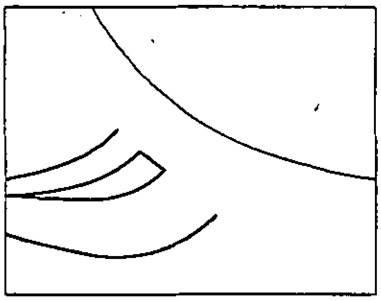


Fig. 9a - Curved diffuser design
($\theta_{in} = 46°$, $\theta_{out} = 46°$, AR = 1.9)

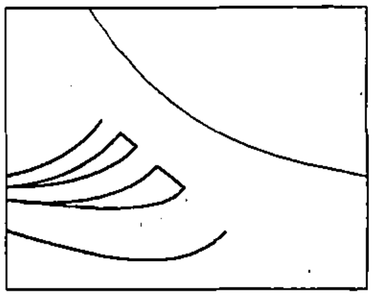


Fig. 9b - Multiple splitter design
($\theta_{in} = 44°$, $\theta_{out} = 53°$, AR = 2.2)

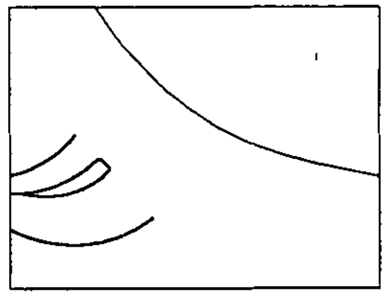


Fig. 9c - Shortened design
($\theta_{in} = 42°$, $\theta_{out} = 51°$, AR = 1.8)

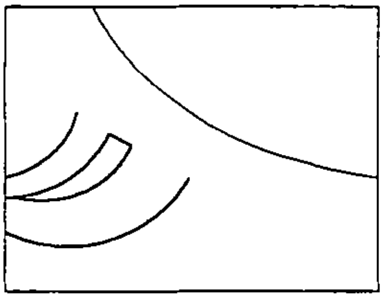


Fig. 9d - Large angled design
($\theta_{in} = 60°$, $\theta_{out} = 70°$, AR = 2.0)

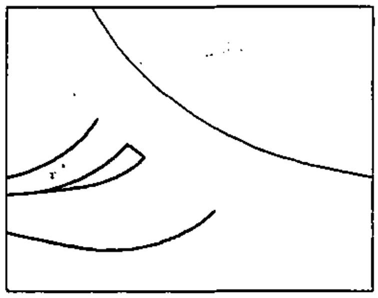


Fig. 9e - Large area design
($\theta_{in} = 46°$, $\theta_{out} = 52°$, AR = 2.5)

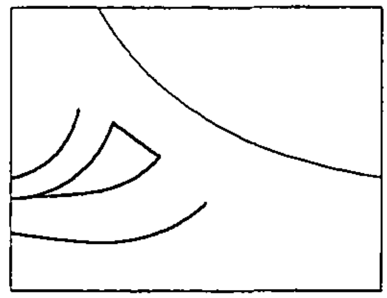


Fig. 9f - Variable angled design
($\theta_{in} = 46°$, $\theta_{out} = 77°$, AR = 2.1)

Table 2 - Turbulence model sensitivity analysis results

% change from standard k-ε model	RNG k-ε model	RSM model
Diffuser loss, K_{diff} (%)	+ 6%	-12%
Diffuser recovery, C_{pr} (%)	- 16%	+7%
Dump loss, K_{dump} (%)	0%	-3%
Overall loss, K (%)	+14%	-24%

The CFD model was run using the three turbulence models available in Fluent/UNS, the standard k-ε model, renormalisation group (RNG) k-ε model and the Reynolds stress model (RSM), and the corresponding mass averaged loss coefficients are given in Table 2.

The results show that the variation in overall loss is quite high, mainly due to the difference in diffuser performance. The RSM model predicts a similar overall flow pattern to the standard k-ε model, but less separation is predicted within the diffuser, probably because the RSM model requires a higher mesh density to correctly calculate the shear stress terms.

The RNG model had convergence problems, since it is more sensitive to separating flows and physical instabilities. Although the loss predictions are closer to the rig results, the separation predictions are pessimistic and the near-wall flow profiles are significantly different to the oil dot flow results. The k-ε model is stable and provides results which are qualitatively good and produces loss coefficients which generally lie between the RNG and RSM results. Hence the k-ε turbulence model was selected for use in the comparative analysis.

The validity of conducting tests using a single sector rig running at ambient conditions compared to the real engine conditions was also confirmed using CFD modelling. The effect of the side walls between combustors reduces the overall loss slightly, due to the more uniform flow through the inner channel. While application of engine conditions (mass flow, pressure and temperature) using the rig inlet profile has little effect.

When an initial turbulence intensity of 20% is used to generate the turbulence profile, the results predict that the diffuser operates more efficiently (higher C_{pr}), with a percentage change of -9% in overall loss. When the inlet velocity profile was adjusted to simulate a measured engine profile, based on tests conducted using an aero diffuser, the diffuser performance drops due to increased separation with a percentage change of +10% in the overall loss. This confirms that accurate results require accurate inlet parameters, but since relative pressure losses are more important when comparing diffuser designs, the effect of inlet parameters should not be significant.

In general, although the actual loss predicted by the CFD model is optimistic, the good flow profile predictions demonstrate that this model and technique can be confidently used as a comparative design tool. Although some sensitivity to inlet conditions exist, by maintaining the same boundary conditions for each analysis, the effect of design modifications on diffuser performance should be realistic.

3D CONCEPT SELECTION

Since the model is quite large, it was not feasible to analyse and optimise each suggested idea in three-dimensions. So, an initial concept selection phase was undertaken, prior to a selective design optimisation. Based on the understanding gained from the datum model, previous experimental work and knowledge, the following possible solutions were put forward.

- Curved diffuser designs to direct and diffuse the flow.
- Vortex controlled diffusers.
- Discharge nozzle modifications to reduce the blockage and interference effects.
- Casing modifications to direct flow and eliminate recirculation.

Although the other ideas are still being considered, this work focused on improving the size and shape of a curved diffuser design, since there was greater confidence in achieving large improvements quickly at lower manufacturing cost and risk.

The aim of the curved diffuser designs is to turn the flow towards the combustor, prevent the existing separation and maintain a high diffuser efficiency. A family of six curved diffuser designs were considered (Figs. 9a to 9f) to investigate how diffuser performance is affected by the main design parameters. In particular, an optimum design will minimise the dump loss, while maximising diffuser efficiency and effectiveness.

The shape of each curved diffuser was generated using a method for designing annular diffusers, which can take into account the effects of compressibility and canted configurations by use of the area bisector line (Adkins, 1983). Each channel was calculated as a separate diffuser and the profile of the area bisector line was formed in

Table 3 - Results for the 3D concept selection

% change from datum CFD results	K (%)	K_{diff} (%)	C_{px} (%)	K_{dump} (%)	Figs.
Curved diffuser	-11.4	+22.5	-5.6	-19.9	9a
Multiple splitters	-9.7	+70.8	+1.9	-24.8	9b
Shortened	+6.6	+44.9	-46.3	-35.4	9c
Large angled	-5.4	+44.9	-18.5	-25.5	9d
Large area	-6.9	+84.3	-24.1	-31.7	9e
Variable angled	-14.6	+34.8	-13.0	-29.8	9f

three sections, each with different radii of curvature (Flaxman, 1998).

This method can only be considered approximate, since the boundary layer development will vary in practice, due to the inlet velocity profile and specific diffuser geometry. However, as a first estimate of the diffuser design and analysis of diffuser types, this provides a fast, simple method for specifying the diffuser design.

The basic curved diffuser design (Fig. 9a) turns the flow through 45°, which represents the flow exit angle from the current in-service diffuser (AR = 2). An additional splitter (Fig. 9b) enables a larger area ratio and spreads the flow out around the discharge nozzle. Shortening the diffuser (Fig. 9c) reduces the blockage, but drops the area ratio; alternatively the blockage could prevent separation and so allow a larger area ratio (Fig. 9d). Based on previous experimental data (Jamieson, 1996), a higher exit angle (Fig. 9e) should turn more flow into the combustor and reduce the dump loss, while a variable angled design (Fig. 9f) may be required to maintain discharge nozzle cooling flow.

Each of the curved diffuser designs (Figs. 9a to 9f) were meshed into a 3D CFD model (including struts), and run at the same boundary conditions and solution parameters as the datum model, and the loss coefficients are compared in Table 3. The results of this analysis were very promising, since most of the designs predicted an improvement in the overall loss. When the flow is angled upwards, less recirculation and re-acceleration occurs around the discharge nozzle, and so the dump loss drops significantly.

The large angled (Fig. 9d) and variable angled (Fig. 9f) designs predict a relatively low dump loss with significant room for improvement in the diffuser performance. So, these two types of diffuser offered the best potential for minimising the overall loss.

2D DESIGN OPTIMISATION

Since a significant amount of time is required to create and solve each 3D CFD model, it was not feasible to employ a 3D optimisation process. The direction and position of the diffuser exit flow remains fairly constant for each type of diffuser, so the discharge nozzle blockage could be considered fixed. Therefore, it was possible to simplify the problem into a 2D axisymmetric analysis.

The optimisation of the two selected curved diffuser designs was conducted using a 2D modelling approach. This method was verified by comparing the 2D model results against a representative slice of the 3D model. The two selected curved diffuser designs were then optimised using a 2D iterative approach, before being converted into a full 3D model, and so determine the predicted improvement in diffuser performance.

2D Model Verification

The CFD model was validated against a slice 10° off the combustor centreline (Cut-2), which should be representative of the flow through the diffuser i.e. midway around the annulus of the 3D CFD model. The 2D diffuser mesh, geometry and boundary conditions were identical to the 3D CFD model. The downstream blockage was then adjusted until the 2D model predictions matched the results on the 3D slice, using the following criteria.

1. Total pressure loss and static pressure recovery coefficients for both channels to check that the 2D model correctly predicts diffuser performance and channel mass flow distribution.
2. Static pressure profile along the diffuser walls, together with the total pressure profile at the diffuser exit, to confirm that the 2D model accurately simulates the diffuser flow profile.

Overall, the results of the 2D validation demonstrate that the diffuser design could be confidently optimised using a 2D model and are within 7% of the 3D predictions. The contours of velocity magnitude in the validated original 2D design for each curved diffuser design are shown in Figs. 10a and 10e.

2D Optimisation Procedure

Once the shape of the discharge nozzle blockage had been determined within the 2D model, the accuracy of the 2D model was improved, by refining the mesh so that the two-layer zonal boundary layer model could be applied.

An optimum diffuser design should maximise the diffuser performance, while maintaining a low loss i.e. minimise insufficient and inefficient diffusion. A secondary requirement, which follows on from these conditions, is to generate a uniform velocity profile at the diffuser exit and eliminate separation.

There are currently few design rules available for curved annular diffusers with a downstream blockage, so an iterative approach was developed. The following parameters were used to study the overall diffuser performance, determine the effect of previous changes on performance, and so enable further modifications to be suggested.

1. Static pressure profiles along all diffuser walls, to provide information on where diffusion occurs.
2. Wall shear stress profiles on the diffuser walls, to determine the likelihood and position of flow separation.
3. Total pressure loss and static pressure recovery coefficients for each channel separately and the whole diffuser, to analyse and compare diffuser performance.
4. Contours of velocity magnitude, combined with the channel mass flow distribution, to investigate the flow spread and channel loading.

When several design changes failed to produce an improvement in diffuser performance, the geometry of the diffuser was considered optimised. This produced a high performance diffuser close to the optimum. Since this is a 2D optimisation process for a 3D problem, there are no perfect optimum designs because 3D variations in the flow and blockage around the annulus will affect the diffuser performance. However, these 3D variations should have a secondary effect, and so the final 2D design should improve the overall diffuser performance.

Prior to a 3D analysis of the optimum design, it was necessary to reduce the number of cells and apply the non-equilibrium wall function. The 2D coarse mesh was then adjusted until the results agreed to within 1.5% of the 2D refined mesh predictions.

The results confirm that the diffuser performance has been significantly improved using the 2D optimisation process. This is due to the more uniform diffuser flow profile, shown in the contours of velocity magnitude for the 2D optimum designs (Figs. 10b and 10f).

3D Optimum Designs

Based on the coarsened 2D optimum diffuser models, 3D CFD models of the optimum diffuser designs were created. The diffuser geometries were swept around the annulus and struts were inserted into the outer channel. All other aspects of the boundary conditions were identical to the in-service datum CFD model, to allow direct comparison of the predicted pressure loss and system performance.

The 3D CFD model results (Table 4) for the optimum diffuser designs confirm that significant improvements in the overall system performance were predicted. This was achieved by reducing the dump loss and improving the diffuser efficiency. Although the absolute losses predicted by the CFD model are considered optimistic, it is fairly certain that large improvements would occur in the engine.

Since the design method involved a 2D optimisation process, it was necessary to verify that the 2D optimum diffuser design results were relevant to the full 3D solution. The 3D diffuser performance predictions on Cut-2 are within 10% of the 2D refined mesh results, which confirms and further correlates the accuracy of the 2D model.

Large angled curved diffuser design

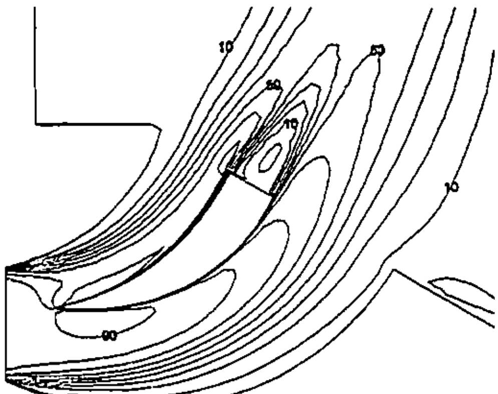


Fig. 10a - 2D Original design

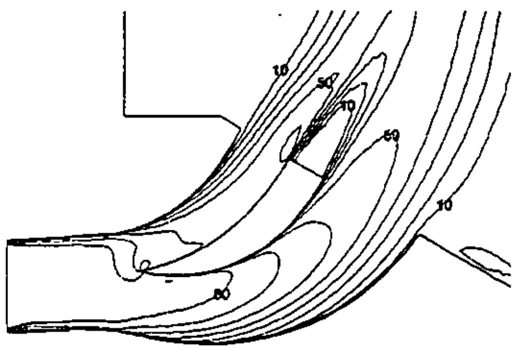


Fig. 10b - 2D Optimum design

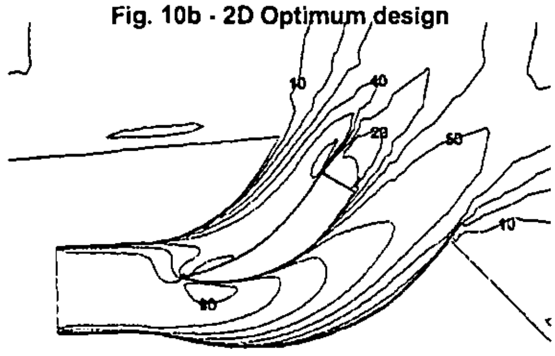


Fig. 10c - 3D Optimum design on cut-2

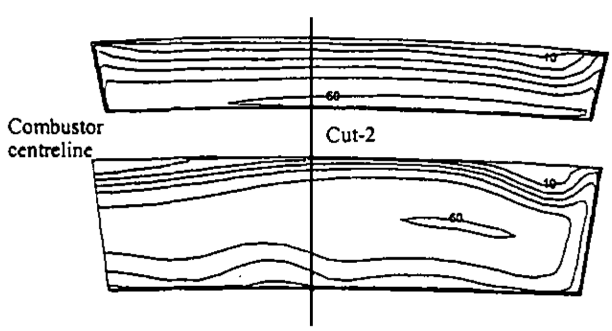


Fig. 10d - 3D Optimum design at diffuser exit

Variable angled curved diffuser design

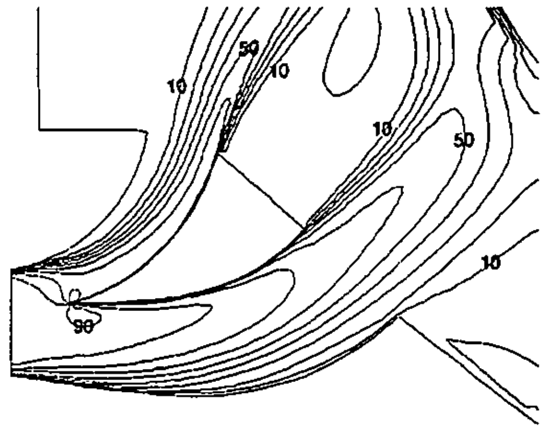


Fig. 10e - 2D Original design

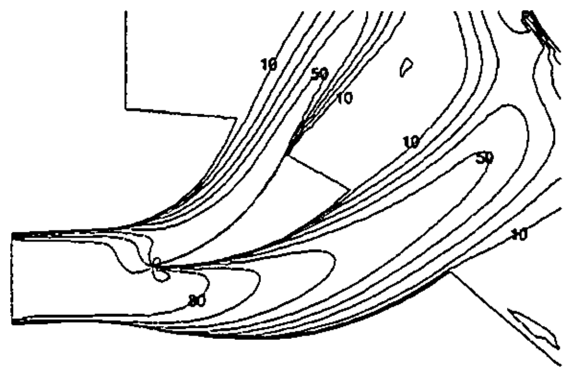


Fig. 10f - 2D Optimum design

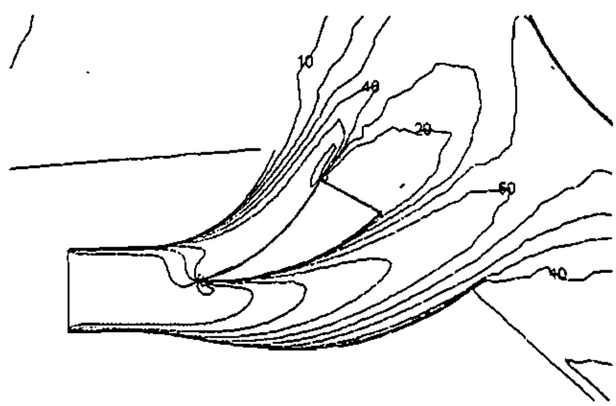


Fig. 10g - 3D optimum design on cut-2

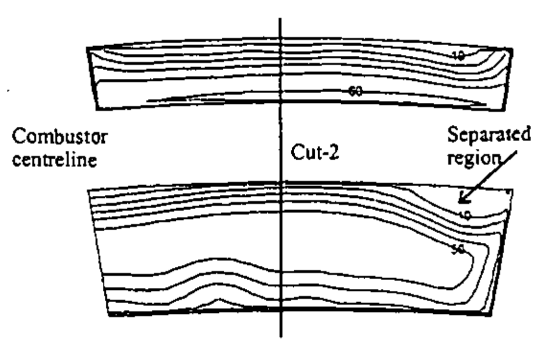


Fig. 10h - 3D Optimum design at diffuser exit

Table 4 - 3D optimum diffuser design results

% change from datum CFD model (Figs. 5b, 5c)	Large angled CFD model	Variable angled CFD model
Figures	10c, 10d	10g, 10h
Diffuser loss, K_{diff}	-26.6	-17.0
Diffuser recovery, C_{pr}	+6.8	+10.1
Dump loss, K_{dump}	-21.1	-16.9
Overall loss, K	-23.4	-23.4

The overall predicted diffuser performance is slightly lower than the performance of the 2D optimum designs, due to variations in the diffuser exit flow profile caused by the circumferentially changing blockage. Figures 10d and 10h show that near the combustor centreline the presence of the discharge nozzle forces the outer wall of the inner channel to separate.

DISCUSSION

Since the predicted CFD pressure losses are considered optimistic, the overall total pressure loss could be scaled to give a better estimate of the actual loss. Based on a constant turbine temperature, the predicted reduction in pressure loss should significantly increase the RB211-DLE engine performance (power output or sfc), making it more attractive to customers at little extra cost.

Although the CFD model predicts a large performance benefit from the optimum diffuser designs, confirmation of these results is required before manufacture. This will involve perspex single sector rig testing of the two best designs, which enable further verification of the CFD model and validation of the design process. The rig will also measure other effects e.g. transient instabilities, noise generation, turbine NGV cooling and discharge nozzle temperatures.

The use of CFD as a proactive design tool has generally been applied to turbine and combustor blades. However, the benefits of using a CFD design approach for other areas of the gas turbine are enormous. The fast turnaround of designs, especially in 2D, combined with the information available within a CFD simulation, reduces project timescales, costs and rig testing schedules.

Since the optimisation process applied to the RB211-DLE diffuser has been successful, this method could be applied to diffusers in other industrial gas turbine engines. Alternatively, the effect of casing and noise attenuation designs on flow profile and performance within the combustor module could be investigated using this CFD model. However, once the confidence and speed at setting up and solving CFD models improves, this technique could be applied to many design problems involving fluid dynamics.

In the future, circumferentially varying diffuser designs could be considered using this approach. This could be achieved by optimising the diffuser at several positions around the annulus, improving the overall diffuser performance and allowing a longer diffuser midway between combustors. This could be combined with casing modifications, which would improve the flow profile by eliminating recirculation.

CONCLUSIONS

The CFD simulation confirms that the overall flow profile, diffuser loss and near-wall velocities are generally well predicted. Although some differences exist in the diffuser flow profile due to inaccuracies associated with the turbulence model and inlet profile. Since the pressure loss predictions are also quite sensitive to inlet profile, this represents an area of concern within the CFD model. However, although the absolute pressure loss predictions are optimistic, the results suggested that this method can be used as a comparative design tool.

The pressure loss predictions suggest that a significant improvement in SFC (based on a constant turbine temperature) would be achieved if the optimum diffuser designs were installed into the RB211-DLE engine. This confirms that the optimisation procedure applied to the RB211-DLE diffuser represents a fast, accurate method of improving the diffuser design. This process involved the preliminary validation of the CFD model against existing data, followed by an initial 3D concept selection phase, before conducting a 2D optimisation of the best ideas. Finally, the optimum design are to be verified against experimental data prior to manufacture.

ACKNOWLEDGEMENTS

The authors gratefully acknowledge the assistance of Rolls Royce for this work and their permission to publish this paper. Thanks also to Fluent Europe Ltd. for their advice and support.

REFERENCES

Adkins, D.C. and Wardle, M.H., 1983, "A Method for the Design of Optimum Annular Diffusers of Canted Configuration," ASME 90-GT-52.

ESDU 75026, 1975, "Performance of Circular Annular Diffusers in Incompressible Flow," ESDU International plc.

ESDU 76027, 1976, "Introduction to Design and Performance Data for Diffusers," ESDU International plc.

Flaxman, J., 1998, "Canted Diffuser designs", Internal report.

Jamieson, J., 1996, "Rig Tests of Curved Diffusers", Internal report.

Miller, D.S., 1990, "Internal Flow Systems," Second edition, BHR Group Limited.

Willis, J.D., Toon, I.J., Schweiger, T. and Owen, D.A., 1983, "Industrial RB211 Dry Low Emission Combustor system," ASME 93-GT-391.