PREDICTION AND MEASUREMENT OF THE TOTAL PRESSURE LOSS IN AN ENGINE REPRESENTATIVE DIFFUSER SYSTEM

A. R. Little, P. A. Denman & A. P. Manners
Department of Aeronautical and Automotive Engineering and Transport Studies, Loughborough University of Technology, Loughborough, Leicestershire, United Kingdom.

ABSTRACT
An experimental and computational investigation of the flow in an engine representative diffuser system has been performed. Many parametric changes to the system were considered including inlet conditions, pre-diffuser geometry, cowl geometry and pre-diffuser strut position. The focus of this paper is an overview of the results from the twelve configurations that were both predicted and experimentally tested.

It was shown that the CFD predictions contained a high degree of numerical error and that to reduce this to a low level would be currently impractical for a parametric study of this type. However, by carefully maintaining approximately the same degree of numerical error in the predictions, it was shown that a valid parametric study could be performed to the extent that the same conclusions about the parametric changes can be drawn equally from the predictions or measurements with one small exception. It was recognised that the level of agreement between the predicted and measured losses was to a degree fortuitous and that the k-ε turbulence model performs poorly for this type of flow.

NOMENCLATURE

\[ A \] area
\[ K \] calibration factor
\[ k \] turbulent kinetic energy
\[ P \] total pressure
\[ P \] mass weighted total pressure
\[ \rho \] density
\[ \rho \] mass weighted static pressure
\[ p_{st} \] measured pseudo-static pressure
\[ \tau \] radial distance from engine centreline
\[ U \] magnitude of mean velocity component
\[ U_i \] mean velocity component
\[ U_n \] mean velocity component normal to surface

\[ u_i \] fluctuating velocity component
\[ V \] volume
\[ \epsilon \] dissipation rate of turbulent kinetic energy
\[ \lambda \] total pressure loss coefficient
\[ \mu \] coefficient of viscosity

INTRODUCTION
The diffuser system of a gas turbine combustion system is the region between the last stage of the compressor and the entry to the combustion chamber. The role of this region is primarily to guide the flow to the various features of the combustion chamber with a minimum loss in mechanical energy and a maximum rise in static pressure (LeFebvre (1983)). Despite a performance penalty, the majority of modern diffuser systems use a dump rather than a fully faired design due to its relative stability, insensitivity to flow distortion and thermal distortion of the walls.
studies of typical diffuser systems (e.g. Fishenden and Stevens (1977)) followed by extensive experimental testing. This is an expensive and time consuming process which is complicated further by the interest in more modern designs to which existing correlations may not be applicable. For example, features such as double annular combustion chambers, passages a large proportion of the air through the cowl and the addition of substantial load-bearing radial struts.

Computational Fluid Dynamics (CFD) potentially offers a considerable improvement to this design process by replacing the existing correlations with a method which can be applied to designs which differ substantially from current practice. It can perform parametric studies more rapidly and cheaply given the initial investment in CFD methods, expertise and computational facilities. In addition, it provides an important and new piece of information which is the reason for the difference in performance of various designs. Although this information tends to be under utilised at present, it should eventually lead to a much better understanding of the flow and a consequent reduction in the number of modifications to be evaluated either computationally or experimentally. However, this is all subject to the CFD method producing sufficiently accurate predictions.

A number of experimental investigations of diffuser systems have been performed. Fishenden and Stevens (1977) investigated the effect of pre-diffuser geometry, dump gap and flow split between the inner and outer annuli for a model dump diffuser system using a fully annular test rig and a solid combustor. Srinivasan et al. (1989) also investigated the effect of varying the flow splits between the inner and outer annuli but included the effect of cowl porosity. Hestermann et al. (1991) investigated the performance of a diffuser system with a solid cowl in order to establish the effects of pre-diffuser geometry on the overall performance. More recent investigations have used slightly more realistic geometries. Studies carried out by Carrotte et al. (1993) have investigated the magnitude and location of the loss of total pressure within the diffuser system of current engine designs.

Computationally, several investigations have been performed although most of the early contributions to the open literature have involved 2D planar or axisymmetric predictions such as Shyy (1985) and Koutmos and McGuirk (1989). These investigations have used a finite volume scheme, body-fitted meshes and the k-ε turbulence model. More recently, several papers have been published which include both CFD and experimental results. Srinivasan et al. (1990) and Carrotte et al. (1993) both showed good agreement for the mean flow field but less favourable agreement for the turbulence quantities and the various performance parameters such as static pressure recovery and total pressure loss. Both of the experimental investigations employed sector rigs. At present there are few contributions in which three dimensional calculations have been performed. Karki et al. (1992) modelled a 45° sector rig in which burner feed arms and struts were included and highlighted the strong three dimensional nature of the flow within the sector.

This paper presents an overview of some of the results from an experimental and computational investigation of the flow in an engine realistic diffuser system. Several parametric changes were made to the system such as inlet conditions, pre-diffuser geometry, cowl geometry and position of pre-diffuser strut relative to the burner feed arm. Over 40 experimental tests were conducted in a period of 15 months and over 40 3D predictions were generated on a small parallel processing computer in a period of about 3 months. The focus of this paper is an overview of the results from the 12 configurations that have been both experimentally tested and predicted.

GEOMETRY

The diffuser system considered was for a double annular combustion chamber which is being increasingly adopted by aero engine manufacturers in order to satisfy future pollutant emissions regulations. The primary focus of this research was to minimise the total pressure loss particularly to the inner and outer annuli although there were several other secondary considerations such as the air feed to the fuel injectors. The experimental investigation was conducted using a perspex model of the diffuser system in which burner feed arms were included but combustor primary ports and cooling rings were omitted. Several geometrical variations of the diffuser system were considered such as alternative pre-diffuser designs, alternative cowl designs and the position of the pre-diffuser struts relative to the burner. These are briefly described below.

Five different pre-diffusers were tested. The first design of area ratio 1.6 was made up of curved walls that turned the flow outward to a cant angle of 11.5° at the exit. The second design was of a similar shape but with an increased area ratio of 2.0 and a thin vane along the centre. The three remaining designs split and turned the flow by varying amounts towards the inner and outer annuli. The area ratios of these designs varied between 1.6 and 2.0. Outlines of all the pre-diffuser designs are shown in Figure 2. A thick load bearing radial strut was present in all the pre-diffuser designs and was placed either directly in-line with or midway between the burners.

![Figure 2: Pre-diffuser Geometries](image-url)

Two cowl configurations were tested. The first comprised a single cowl covering both sections of the combustor with a plunged elliptical slot cut out for the burner arm. The second consisted of separate cowls for each section of the combustor with round plunged holes for the injectors. Outlines of both cowl
configurations are shown in Figure 3.

Figure 3: Cowl Geometries

Figure 1 shows the first pre-diffuser with the strut in line with the burner and the single cowl. This was the datum configuration.

CFD METHOD

With the exception of the grid generation and parallelization, the CFD methods used for this set of predictions can be viewed as currently ‘standard’ methods which are available in several commercial codes. They are very briefly summarised below.

The fluid motion was governed by the incompressible, constant density, isothermal, time-averaged form of the Navier-Stokes equations. The co-ordinates of the differential operators were expressed in general non-orthogonal co-ordinates whilst the velocity and stress components were expressed in Cartesian co-ordinates. The equations were discretised using a collocated finite volume method. Pressure/velocity decoupling was prevented by introducing “pressure smoothing” in the manner of Rhie and Chow (1982). All terms were evaluated using centred differencing except for the convection terms which were evaluated using either hybrid differencing (Spalding (1972)) or QUICK differencing (Leonard (1979)). The algebraic equations were solved using the SIMPLE algorithm of Patankar and Spalding (1972). The turbulence was modelled with the k-ε model of Jones and Launder (1973) using the standard set of empirical constants as optimised by Jones (1980). The code has been adapted to run in a fully implicit manner on distributed memory parallel processing computers.

The mesh used by the CFD code was generated as follows. Firstly, a CAD solid model of the diffuser system was specified from a set of engineering drawings. A mesh was algebraically generated on the surface of the solid model and subsequently smoothed by solving a set of elliptic partial differential equations. Figure 1 shows an example of the grid on the solid surfaces within the solution domain. The internal mesh was then generated by using the surface mesh as boundary conditions for a set of Poisson equations as described by Thompson et al. (1985). Further details on the grid generation are given in Manners and King (1991).

The CFD predictions were of the tested geometry and not an equivalent engine configuration. Two simplifications were made; the circular rings of holes in the fuel injectors were replaced with an annular gap of equivalent area and the plunging on the cowl was omitted. The first was due to lack of grid resolution. The second was due to ambiguities in the mathematical representation of plunged holes in general and, furthermore, it was believed unnecessary to fully represent this region given that the primary interest of the study was in the losses to the inner and outer annuli. The flow splits would not be affected by the change in discharge coefficient because they are imposed by the exit boundary conditions.

For the predictions, two sets of inlet conditions were used: fully developed and a set based on some measurements at the inlet. The early predictions all used fully developed inlet conditions because measurements at the inlet had not been taken. In order not to invalidate the various parametric studies, many of the later predictions also used fully developed inlet conditions.

The set of fully developed conditions was obtained by performing a separate upstream prediction. The measured data at the inlet consisted of the axial velocity component shown in Figure 4 plus a free stream turbulence intensity measurement of 4%. Again, a separate upstream prediction was performed in order to approximately match the measured inlet data and obtain reasonable values for the solution variables not measured.

The downstream boundary conditions were specified as zero gradient normal to the exit for all solution variables except pressure which was extrapolated from internal values. The mass flow splits imposed on the 5 downstream annuli were those used experimentally and are shown in Figure 5.

Since there was no swirl in the model fuel injector, symmetry conditions could be used on a half sector width as shown in Figure 1. Wall functions were used to impose “law of the wall” conditions for cells adjacent to walls as described in Manners (1988).
EXPERIMENTAL METHOD

The experimental data were recorded under cold conditions from a 48° 4 burner sector perspex model of a double annular combustor system. The test rig shown in Figure 6 was vertically mounted and supplied with air from a centrifugal fan via an under-floor plenum. Air entered the rig through a flared intake section containing a total of 19 thin uncambered blades positioned immediately upstream of the model working section. The blade thickness/chord and space/chord ratios were 0.08 and 0.38 respectively, and the trailing edge of each was located at traverse plane A. Although the vanes were aerodynamically unloaded, their presence generated modest blade wakes that were allowed to pass freely into the model working section. The normalised contours of axial velocity recorded at the inlet reference plane A are shown in Figure 4. Turbulence grids could also be added to the inlet section in order to generate higher levels of turbulence intensity at the inlet reference plane. The turbulence intensity at the mid-passage position immediately upstream of the vanes could be increased from approximately 1% to 4% of the mean inlet velocity. Although the grid generated turbulence was unrepresentative of that present immediately downstream of a compressor, its inclusion allowed the effect of a change of turbulence to be determined.

![Figure 6: Experimental Test Facility](image)

The test rig working section consisted of a pre-diffuser followed by a double annular combustor system, comprising main and pilot combustors surrounded by inner, outer and splitter annuli. The system was outwardly canted and the modular nature of the working section allowed easy installation of alternative pre-diffuser and combustor cowl configurations. Four burner feed arms were also included in the combustor test section. However, the burner rings (i.e. swirlers and fuel injectors) to which these were mounted were not engine representative but served only to provide the correct combustor backplate porosity.

All of the experimental data were recorded at an inlet Mach number of 0.26 ± 0.2% and this operating condition corresponded to a blade chord Reynolds number of 2.1 x 10^5. The mass flow rate through each combustor system element was controlled by butterfly valves located in the exhaust pipes approximately 30 pipe diameters downstream of the test section. Using these devices, the mass flow rate through each annulus could be controlled to within ±0.1% of the total mass of air passing through the test rig.

The performance of the diffuser system was derived for each configuration from time averaged pressure measurements recorded at the 5 traverse planes shown in Figure 6. The data were measured relative to the wall static pressure present in the intake section upstream and recorded using Furness FCO 44 differential pressure transducers. Full area traverses were performed using a "button hook" probe (see Figure 7) at pre-diffuser inlet (Plane A), pre-diffuser exit (Plane B) and at entry to both the inner (Plane C) and outer (Plane E) annuli. The axial location of planes C and E coincided with that of the primary port admission holes that would be present in an engine. The splitter annulus (Plane D), however, could not be traversed in the same way. Instead, total pressure data were recorded by means of 6 fixed pitot rakes, equally distributed across the central burner sector of the model. Each pitot rake was made up of 6 pitot tubes and the static pressure distribution was inferred from static tappings placed in the walls of the splitter annulus.

At pre-diffuser entry, an area traverse was performed across a single blade space in order to define the experimental inlet conditions. The result is presented in Figure 4 in terms of contours of axial velocity. Area traverses were also conducted for each configuration at planes B, C and E but were confined to the central burner sector (i.e. ±6°) of the 4 burner sector model in order to minimise the influence of the side walls. The adverse effects of sector endwall mass flow displacements were recognised and have previously been addressed experimentally by Carrote et al. (1993) and computationally by Little and Manners (1993).

The area traverses at planes A, B, C and E were conducted using a "button hook" probe, the principle of which is indicated in Figure 7.

![Figure 7: Button Hook Probe Operation](image)

The technique was adopted by Carrote et al. (1993) to evaluate the relative performance of dump and short faired combustor diffuser systems. The device is operated in two modes. The first as a conventional pitot probe to record the upstream total pressure and the second to obtain a "pseudo" static pressure, with the probe head rotated through 180° to face the downstream direction. Using a suitable calibration factor $K$ (assumed constant), the true local flow field static pressure can be deduced from:

$$K = \frac{P - P_{ps}}{P - p}$$
where $P$ and $p_{ps}$ are the measured total and "pseudo" static pressures respectively and $\rho$ is the true local static pressure. Probe calibration was performed over the desired velocity range in a low turbulence environment in which the conditions could be accurately defined. As a result, the calibration constant, $K$, could be determined to within $\pm 1\%$ of the mean value. After calibration, a probe of this type can be used to investigate flow fields in which the presence of flow curvature prohibits the use of pitot tubes and wall static tappings only.

The mass weighted total pressures recorded using the button hook probe at traverse planes A, B, C and E could be repeated to within $\pm 1\% \text{mmH}_2\text{O}$, a variation of less than $\pm 0.25\%$ of the dynamic head measured at the inlet reference plane A. This translates to an error of $\pm 0.003$ in the derived mass weighted total pressure loss coefficient at each traverse plane.

**LOSS PARAMETERS**

The loss between two stations 1 and 2 can be determined by integrating the mass-weighted total and static pressures:

$$\lambda = \frac{P_2 - P_1}{P_1 - p_1}$$

where

$$P = \frac{\int (p + \frac{1}{2} \rho U^2) \rho U_n dA}{\int \rho U_n dA} \quad \text{and} \quad p = \frac{\int p U_n dA}{\int \rho U_n dA}$$

This is subsequently referred to as the "flux" method and was used for both the measurements and the predictions.

The predictions can also evaluate the loss by integrating throughout the volume of interest the work done by the mean flow against the viscous and turbulent stresses:

$$\lambda = \int_{\text{inlet}} \left( -\rho u_i u_j + \mu \frac{\partial U_i}{\partial x_j} \frac{\partial U_j}{\partial x_i} \right) dV$$

This is subsequently referred to as the "volume" method. The difference between the two methods of loss evaluation is a sensitive measure of the numerical error in the solution. Further discussion is given in Little and Manners (1993).

**RESULTS**

**Grid Refinement**

Predictions were performed for the datum geometry (i.e. pre-diffuser 1, single cowl and strut in-line with the burner) shown in Figure 1 with fully developed inlet conditions for two levels of refinement; a coarse grid of approximately 70 000 grid points and a medium grid of approximately 450 000 grid points. The main flow features for both predictions were similar although the medium grid had significantly better resolved shear layers.

The level of numerical error present in the predictions was estimated by comparing the overall loss coefficients evaluated by the two alternative methods described and is shown in Figure 8.

![Figure 8: The Effect of Grid Refinement](#)

For the coarse mesh, the overall loss evaluated by the "flux" method is approximately double that of the "volume" method indicating a large level of numerical error. The level of agreement is better for the medium grid density but the prediction clearly cannot be considered to be approaching grid independence. A further doubling of the grid density would require nearly 4 million grid points but was not possible with the small parallel processing computer available for the predictions. Subject to the flow pattern remaining unchanged, such a prediction would enable a good estimate of the grid independent overall loss to be made. See Little and Manners (1993) for further discussion.

All the predictions reported below used the coarse grid density in order to perform a parametric study in a reasonable time. Care was taken to maintain approximately the same grid density for all geometries in an effort to have similar levels of numerical error in all the predictions.

**Measurements and Predictions for the First Set of Inlet Conditions**

The measurements were taken without an upstream turbulence grid giving a free stream turbulence intensity of 1% and an inlet axial velocity profile similar to that shown in Figure 4. The predictions were performed before the inlet conditions were measured and in their absence fully developed conditions were specified. Given this consistent difference between the measurements and predictions, seven configurations can be compared and are summarised in the table below:

<table>
<thead>
<tr>
<th>Configuration</th>
<th>Pre-Diffuser</th>
<th>Cowl</th>
<th>Strut/Burner</th>
</tr>
</thead>
<tbody>
<tr>
<td>A1</td>
<td>1</td>
<td>single</td>
<td>out-of-line</td>
</tr>
<tr>
<td>A2</td>
<td>3</td>
<td>double</td>
<td>in-line</td>
</tr>
<tr>
<td>A3</td>
<td>3</td>
<td>double</td>
<td>out-of-line</td>
</tr>
<tr>
<td>A4</td>
<td>1</td>
<td>double</td>
<td>in-line</td>
</tr>
<tr>
<td>A5</td>
<td>2</td>
<td>in-line</td>
<td></td>
</tr>
<tr>
<td>A6</td>
<td>4</td>
<td>double</td>
<td>in-line</td>
</tr>
<tr>
<td>A7</td>
<td>5</td>
<td>double</td>
<td>out-of-line</td>
</tr>
</tbody>
</table>

Table 1: Configurations for First Set of Inlet Conditions
Figure 9 shows a comparison between the predicted and measured total pressure loss in the outer, splitter and inner annuli (planes C, D and E respectively in Figure 6) with respect to the inlet plane A. The predicted and measured losses were evaluated using the "flux" method.

Given the high levels of numerical error, there is surprisingly good agreement between the measurements and predictions and the trends, briefly discussed below, have been well predicted. This suggests that the CFD method is successfully modelling the relative change in the balance of the dominant loss generating processes.

The overall level, rather than the trends, is fortuitously close to the measurements. For example, if it were possible to split the overall loss evaluated using the "volume" method into components in Figure 9 then the level of the predicted loss curves would be roughly halved. There are several known deficiencies in the predictions which have combined to bring about the close agreement in the level. The fully developed profile has increased the loss in the inner and outer annuli relative to that in the splitter due to the relative change in total pressure at the inlet feeding the annuli. The high levels of diffusive numerical error have raised the predicted losses (Little and Manners (1993)). The k-ε turbulence model has two significant failings for this type of flow. Firstly, it overpredicts loss in impingement regions (Craft and Launder (1991)) and, secondly, it cannot predict the large increase in loss due to streamline curvature of the flow over the head of the combustor (Bradshaw (1973)). The second failing is almost certainly the larger and leads to an underprediction of the loss in the inner and outer annuli for a fully resolved prediction.

Both the predicted and measured results show the beneficial effects of increasing pre-diffuser area ratio provided that the flow in the pre-diffuser remains attached. The pre-diffusers with the larger area ratios (i.e. 2 and 4 in Figure 2) both have reduced losses to the inner and outer annuli. For these two geometries, the velocity of the flow in the dump region has reduced, reducing the generation of turbulence and hence the losses. This can be seen in Figure 10 which compares the levels of turbulent kinetic energy in the outer dump region for configurations A3 and A6.

The low loss in the inner annulus for configuration A6 was caused by the faired nature of the geometry effectively extending the diffusion length and eliminating the dump recirculation as shown in Figure 11.

The loss between the inlet and the splitter annulus is dominated by the blockages present. There is no blockage present for pre-diffuser 1 with the double cowl (configuration A4) and the loss is consequently small. The boundary layers on the vane of pre-diffuser 2 introduce a significant loss (configuration A5) although this is less than that introduced by the single cowl (configuration A1). The largest loss is caused by splitting and turning the pre-diffuser flow towards the inner and outer annuli (configurations A2, A3, A6 and A7).

When this comparison was initially performed configuration A7 stood out because the predicted losses were significantly larger than the measured losses. An investigation of the predicted flow field revealed a separation on the outer wall at the inlet to the pre-diffuser. This was caused by the coarse grid creating a sharp corner in place of the smooth but tightly radiused corner shown in Figure 2. A few additional axial grid lines eliminated the problem by improving the definition of the pre-diffuser wall curvature. This is an example of one of the main concerns in a computational
exercise using coarse grids, namely, if the flow pattern changes significantly with increased grid resolution, trends will not be properly predicted. In all of the other predictions and measurements, no separation was present in the pre-diffuser.

**Measurements and Predictions for the Second Set of Inlet Conditions**

The measurements were taken with an upstream turbulence grid giving a free stream turbulence intensity of 4% and the inlet axial velocity profile shown in Figure 4. The predictions were performed with a set of inlet profiles approximating the measured inlet conditions. The predictions and measurements can be compared for the five configurations given in the table below.

<table>
<thead>
<tr>
<th>Configuration</th>
<th>Pre-Diffuser</th>
<th>Cowl</th>
<th>Strut/Burner</th>
</tr>
</thead>
<tbody>
<tr>
<td>B1</td>
<td>2</td>
<td>single</td>
<td>in-line</td>
</tr>
<tr>
<td>B2</td>
<td>2</td>
<td>double</td>
<td>in-line</td>
</tr>
<tr>
<td>B3</td>
<td>4</td>
<td>single</td>
<td>in-line</td>
</tr>
<tr>
<td>B4</td>
<td>4</td>
<td>double</td>
<td>in-line</td>
</tr>
<tr>
<td>B5</td>
<td>4</td>
<td>double</td>
<td>out-of-line</td>
</tr>
</tbody>
</table>

**Table 2: Configurations for Second Set of Inlet Conditions**

Figure 12 shows a comparison between the predicted and measured total pressure loss in the outer, splitter and inner annuli (planes C, D and E respectively in Figure 6) with respect to the inlet plane A. Again, the trends have been reasonably well predicted. The effects of cowl and pre-diffuser geometry on the splitter loss are well predicted and similar to the previous set of tests. The effect of changing the pre-diffuser geometry is also well predicted for the inner and outer annuli. The small increase in loss due to moving the strut and burner out of line is not well predicted for the inner and outer annuli although the sign is correct. The relatively small change in loss in the inner and outer annuli caused by changing the cowl is incorrect in the predictions. Although the reason is uncertain without further experimental data, the likely causes are the missing plunging since there is a small amount of spillage out of the elliptical hole in the single cowl and/or the problems with the k-ε turbulence model in impingement regions.

By comparing the measured losses for configurations A5 / B2 and A6 / B5 it can be seen that increasing the free stream turbulence from 1% to 4% has significantly reduced the losses in the diffuser system (see Figure 13). This is caused by the increased levels of turbulence in the pre-diffuser improving the shape of the boundary layers and consequently improving the performance of the pre-diffuser (i.e., flatter exit velocity profile and higher pressure recovery) leading to reduced velocity gradients in the dump region and lower losses. This is in agreement with previous experimental investigations (see Stevens and Williams (1980)).

For the predictions, the comparison between configurations A5 / B2 and A6 / B5 is more complex. In changing from the fully developed to the experimental inlet profile, the velocity profile has changed from being significantly peaked to being largely flat and the overall level of turbulence at the inlet has reduced for the flow entering the inner and outer annuli instead of increasing as in the measurements. The effect of the velocity profile is to decrease the loss in the inner and outer annuli and increase the loss in the splitter annulus due to the increased total pressure at the inlet for the flow entering the inner and outer annuli at the expense of that for the splitter annulus. The effect of the reduced level of turbulence in the inner and outer annuli, as shown by the measurements, is to increase the loss. These two effects combine to leave the predicted loss in the annuli largely unchanged as shown in Figure 13.

**Figure 12: Losses for the Second Set of Inlet Conditions**

**Figure 13: The Effect of Changing the Inlet Conditions**
CONCLUSIONS

The CFD predictions were shown to have a high level of numerical error and that using a sufficiently refined grid to reduce this to a low level would be currently impractical for a parametric study.

By carefully maintaining approximately the same degree of numerical error in the set of predictions, it was shown that a valid parametric study could be performed to the extent that the same conclusions about the parametric changes can be drawn equally from the predictions or measurements with one small exception. The small effect on loss of changing the cowl was incorrectly predicted for the inner and outer annuli and is believed to be caused by the missing plunging of the elliptical hole in the single cowl and/or the deficiencies in the k-ε turbulence model.

It was recognised that the level of agreement between the predicted and measured losses was to a degree fortuitous. Refining the grid would decrease the predicted loss and increase the difference between the predictions and measurements. The turbulence model is the cause of this discrepancy and, along with methods of efficiently increasing the grid resolution, is the subject of further study.

ACKNOWLEDGEMENTS

The work described in this paper was carried out within the Rolls-Royce University Technology Centre at the Department of Aeronautical and Automotive Engineering and Transport Studies, Loughborough University. The authors would like to acknowledge the financial support of Rolls-Royce plc. and the Defence Research Agency (Pyestock) under contracts B1FI-309DC and B1FI-291DC.

REFERENCES


