Separated flow in a three-dimensional diffuser: preliminary validation

By E. M. Cherry, G. Iaccarino, C. J. Elkins and J. K. Eaton

1. Motivation and objectives

In gas turbine engines, the final stage of air compression occurs within the annular diffuser just upstream of the combustor. This component must satisfy conflicting goals of recovering kinetic energy exiting the compressor while supplying reasonably uniform flow and consistent mass splits into the various sections of the combustor. The key challenge in designing the diffuser is to make it as short as possible while avoiding any possibility of massive flow separation. Pressure losses due to the separated flow reduce engine performance while unsteadiness and recirculating flow associated with separation can cause catastrophic engine failure. An optimal design probably operates very near separation for some part of the engine’s operating envelope. Accurate design analysis tools are needed to find the optimum and to avoid unexpected failures during prototype testing.

Previous experimental investigations (Obi et al. 1993; Buice & Eaton 2000) considered a planar geometry and provided a large amount of measurements both in terms of mean velocity and turbulent quantities. In order to guarantee the two-dimensionality of the flow, a very high aspect ratio duct was considered. Durbin (1995) and Iaccarino (2001) performed Reynolds-Averaged Navier-Stokes (RANS) simulations of the diffuser and concluded that good overall agreement with the experiments was obtained using the V2F turbulence model; discrepancies were observed in the recovery region (after the flow reattachment). Additional numerical studies carried out using Large-Eddy Simulations (LES) were conducted by Kalthenbach et al. (1999) and later by Wu et al. (2006); the agreement was again satisfactory although in the region downstream of the separation proved to be the most difficult to reproduce numerically. It was hypothesized that the flow in this region is characterized by long-time unsteadiness and potential flow three-dimensionality. The difficulty in exactly defining the flow conditions in the direction perpendicular to the diffuser plane might prevent reproduction of the experimental configuration in the simulations.

The objective of the present work is to complement the above mentioned study by performing experiments and simulations of a truly 3-D diffuser with simple and well-specified boundary conditions. The experimental setup is designed to provide a challenging test case for numerical models: it involves a well-defined 3-D recirculation region, and a considerable amount of data are collected at realistic Reynolds numbers. In addition, the effect of a small change in the expansion ratio is used to evaluate the ability of the numerical methods to predict trends and sensitivity to the geometry. Current measurements are obtained using Magnetic Resonance Velocimetry (MRV) (Elkins et al. 2003). Simulations are based on a novel unstructured-grid method developed for high-fidelity LES (Mahesh et al. 2004; Ham & Iaccarino 2004). Predictions using RANS turbulence models are also considered to identify the limitations of conventional turbulence models.

An additional objective of this work is to develop a procedure to objectively validate the numerical predictions in a case where detailed volumetric measurements exist. In
the present situation, the experiments consist of three-component mean velocity vectors within the entire diffuser volume. This makes classical comparisons based on the extraction of few selected velocity profiles possibly insufficient (although still useful).

2. Experimental setup

The working fluid for all of the experiments was water. A gadolinium-based contrast agent (Omniscan, Nycomed, Inc.) was added to the water in a concentration of 0.5%. The schematic of the recirculating flow loop is shown in Fig. 1. A centrifugal pump (Little Giant model no. TE-6MD-HC) circulated water at a flow rate of 20.3 L/min. The average volume flow rate was measured using a Signet Scientific MK315.P90 paddle wheel flow meter, which was calibrated using the bucket and stopwatch method described by Elkins et al. (2003) with an estimated uncertainty of 5%. The pump was placed approximately 3 meters from the magnet, and no other metallic parts were used in the loop to avoid influencing the signal detected by the magnetic resonance imaging (MRI) system. Flexible plastic tubing with an inner diameter of 25.4 cm was used to complete the flow loop.

Figure 2 is a photograph of the transition piece, development channel, and test diffuser. The diffuser itself was preceded by three inlet parts made of Plexiglas and stereolithography (SLA) resin. The SLA pieces were fabricated with a normal resolution of 100 μm by Mr. Frank Medina of Keck Laboratory at the University of Texas El Paso.

The upstream transition piece was designed to smoothly morph the cross-section of the flow from a 25.4 cm diameter circle to a rectangle with the same dimensions as the development channel. This section included three sets of grids with 2 mm square holes and a 60% open area to keep the flow from separating and provide uniform mean flow.

Figure 1. Sketch of the experimental setup.

Figure 2. The development channel, the diffuser, and the outlet transition duct as fabricated using stereolithography manufacturing (see Fig. 1)
and turbulence at the development section inlet. The 60-cm-long development channel had a constant rectangular cross section of height 1 cm and aspect ratio 1:3.33. Three grids were included at the upstream end of the development section to achieve a greater flow uniformity. Velocity data a few centimeters upstream of the diffuser inlet showed that the flow was fully developed by the end of this channel.

The test diffuser is attached directly to the development channel exit. Diffuser 1 has a rectangular inlet of height 1 cm and aspect ratio 1:3.33 and a 4 cm square outlet, giving an area expansion ratio of 4.8. The diffuser is 15 cm long. One side wall expands at an angle of 25.6 degrees, and the top wall expands at an angle of 11.3 degrees. The other two walls are straight. Diffuser 2 is also 15 cm long and has the same inlet as Diffuser 1, but its outlet is 4.51 cm x 3.37 cm, giving an area expansion ratio of 4.56. The top wall of Diffuser 2 expands at an angle of 9 degrees and its side wall expands at an angle of 4 degrees. The Reynolds number in both diffusers based on the height of the inlet channel is set to approximately 10,000. Different outlet transition sections are used for the two diffusers because it is necessary to match the dimensions of the diffuser outlet. Both outlet transitions have 10 cm of constant-cross-section channel and then a 10-cm contraction into a circular outlet 25.4 cm diameter.

Velocity data are collected using the method of magnetic resonance velocimetry (MRV) described by MRV uses magnetic resonance imaging to measure the three-component mean velocity vectors in a three-dimensional volume.

All experiments were performed at the Richard M. Lucas Center for Magnetic Resonance Spectroscopy and Imaging at Stanford University. A 1.5-T MR system (GE Signa CV/I, Gmax = 40 mT/m, rise time = 268 ms), with a single channel, head receive coil was used. Data were collected with a sagittal slab 74 mm thick in the spanwise direction and a field-of-view (FOV) of 24 cm in the streamwise and cross stream directions. The imaging matrix was 74 × 256 × 256 yielding 1 mm resolution in the spanwise direction and 0.9 mm

![Figure 3. Geometrical details of the two diffuser configurations used in the present study.](image)
in the streamwise and cross stream directions. A total of 16 complete scans of the flow were averaged to produce the final velocity field.

The mean velocity uncertainty was estimated to be less than 10% following the analysis of Elkins et al. (2003) and Elkins et al. (2004).

Velocity field data were processed using Matlab. The coordinate system was rotated and translated to match the coordinate system of the Solidworks model of each diffuser. The data were then averaged in the streamwise direction using a five-point Gaussian filter.

3. Numerical setup

As discussed in the previous section, the diffuser geometry was designed to be easily defined; the presence of planar surfaces and simple rectangular inlet and outlet ducts facilitates the construction of a very high-quality hexahedral-only mesh. The grid is clustered at the walls to provide good resolution of the near-wall region. In Fig. 4 several cross-sections are reported in the region of the diffuser. The computational domain also includes the development duct and the outlet region (as defined earlier). Most of the calculations reported herein are obtained on a mesh consisting of 1.8 million grid cells (referred to as the coarse grid); an additional mesh (fine grid) consisting of 14 millions cells is also considered for the LES simulations. This is obtained by splitting each hexahedral in the coarse grid into eight elements.

Boundary conditions are simple no-slip walls, velocity inlet and outflow. At the inlet plane, a constant velocity matching the experimental mass flow rate is specified. No fluctuations are added, as the inflow plane is far from the diffuser region. The outlet is treated as a classical convective outlet.

The numerical simulations using LES are performed using CDP, a parallel, unstructured code for accurate flow simulations (Mahesh et al 2004; Ham & Iaccarino 2004). The incompressible Navier-Stokes equations are solved on general polyhedral meshes using a fractional-step procedure. Second-order symmetric discretization in time and space is used. This results in an algorithm that conserves kinetic energy, and introduces no numerical dissipation. The subgrid scale Reynolds stresses are modeled using the dynamic Smagorinsky closure.

In addition to the LES, computations using a commercial software (Fluent) are considered to evaluate the predictive abilities of conventional RANS turbulence models. Several models have been tested, but only the results using the $k-\omega$ SST model (Menter 1994) are presented herein. Results obtained with other eddy-viscosity models are comparable.

The LES computations are performed using unity CFL; this results in a timestep approximately equal to 0.00015 seconds on the coarse mesh. The simulation has been run for approximately 25,000 timesteps to achieve statistical convergence. Afterward the flowfield has been averaged for about 2.5 seconds (or equivalently 15,000 timesteps). On the other hand, the computations on the fine grid have been averaged only for about 8,000 timesteps and have not achieved sufficient convergence of the velocity statistics. The preliminary results are presented in the following section although the computations are ongoing. Calculations on the two grids have been carried out using 128 processors of the ALC Linux cluster at Lawrence Livermore National Laboratory.

Fluent computations are carried out solving the steady RANS equations using 16 processors and have been completed only on the mesh consisting on 1.8 million grid elements.
4. Quantitative comparisons

Comparisons between numerical predictions and measurements are reported in Figs. 5 and 6. The development of the flow in the diffuser is illustrated in Fig. 5 in terms of streamwise velocity isocontours in axial cross-sections. The experiments show the presence of a 3D separation that originates in the top-right corner and then propagates to become almost 2D in the upper part of the diffuser. The RANS simulations strongly overpredict the strength of the separation and result in a flow that is recirculating well in the outlet region. In addition, the flow is separated on the right side of the diffuser as opposed to the measurements that show a recirculation area on the top wall. Mass conservation implies strong acceleration in the left part of the diffuser, resulting in very different velocity distributions in the cross-sections. On the other hand, the LES computations show a much better agreement with the data, especially for the fine grid, although the temporal averages are not converged. In Fig. 6, the streamwise velocity in a longitudinal plane at the mid-span of the development channel cross-section is reported. This plot can be compared directly to the results presented in the 2D diffuser simulated in Iaccarino (2001). The results confirm that the RANS predictions severely over-estimate the strength of the recirculation, while the LES (especially on the fine grid) appear to be in reasonable agreement with the experiments.

In order to provide a more quantitative analysis of the predictions and measurements, six velocity profiles are extracted from the longitudinal plane reported in Fig. 5. The comparisons are reported in Fig. 7. The experiments clearly show a large separated region on the top wall of the diffuser, but the detachment point is approximately located at \( x = 8 \text{mm} \) downstream of the diffuser inlet. This is in strong contrast with the predictions of both RANS and LES, which result in a separation occurring almost immediately at the diffuser inlet. The LES predictions on the fine grid, however, appear to be in much better agreement with the measurements. All of the numerical simulations predict a thinner boundary layer on the bottom wall compared to the measurements.
Figure 5. Streamwise velocity isolevels in four cross-sections along the diffuser 1. The columns represent sections at $x = 4\text{mm}, 8\text{mm}, 12\text{mm}, \text{and } 16\text{mm}$ from left to right (the origin of the $x$-coordinate is fixed at the diffuser inlet).

5. Ongoing and future work

The experimental and numerical results presented in this report have been collected during the first part of this research activity. Current experimental work is focused on the characterization of the second diffuser geometry. On the numerical side, the predictions obtained on the fine grid are considered to be preliminary because the flow statistics have not been averaged for sufficient time. The simulation is ongoing. The computational grid for the second diffuser has been generated but the flow computations have not yet started. In addition to completing these simulations, we intend to perform simulations using a Reynolds stress model to evaluate the limitations introduced by the eddy viscosity approximation.
Another important aspect of the present joint experimental and numerical study is the development of objective measures of the agreement between the datasets. The availability of massive amounts of measurements (in the present case about 400,000 velocity vectors in the diffuser region) makes conventional comparisons such as the one presented in Fig. 7 limited and potentially misleading, as they only focus on a limited region of the domain. In particular, the last profile in Fig. 7 seems to suggest a potential difference in the overall mass flux through the diffuser. In practice, the overall mass flow has been verified to be conserved and in close agreement between experiments and simulations. Each velocity profile in Fig. 7 is a 1D sample of a strongly 2D flow in the corresponding cross-section; mass conservation, on the other hand, is a scalar measure of the flow characteristic in the cross-section. In an attempt to capture the entire flow as a 3D entity, the probability density functions of the velocity vectors are reported in Fig. 8; this represents a global measure of proximity between measurements and numerical predictions although it is only qualitative because by construction it includes a scaling (the integral of the PDF is unity). Figure 8 shows the mean velocity statistics in the recirculating region and the much better agreement of the LES predictions as compared to the RANS.
one (although only the coarse grid results are reported here for the LES). In addition, the PDF of the vertical velocity illustrates a strong asymmetry, while the cross-flow velocity is nearly Gaussian. Future work will focus on ways to compare quantitatively the predictions based on distribution functions and as well as determine local comparison metrics.

Another important component of the validation process is the determination of the uncertainties. Although the experimental characterization has been performed (following what was done earlier in Elkins et al., 2003), we have not included uncertainty bars in the present data as they are not yet available for the computational results. Future work will address this aspect of the numerical simulations.

Acknowledgments

This work is supported by the ONR and the Department of Energy within the Advanced Simulation and Computing Program.

REFERENCES

Figure 8. Probability density function of the three velocity components within the diffuser region. Solid lines are the experimental data, dotted and dashed lines are the RANS and the LES predictions respectively (on the coarse mesh).


