N88-11157

FLOW FIELD MEASUREMENTS IN A 90 DEGREE TURNING DUCT

Roger A. Crawford and Carroll E. Peters
The University of Tennessee Space Institute

NASA Grant NAG3-617

INTRODUCTION

Ducted three-dimensional flows are found in many engineering applications and present a difficult challenge to computational fluid mechanics codes. The three-dimensional development of viscous shear layers has a strong influence on the complete flow field. Some of the most complex flows are found in turbine engines and rocket motor turbomachinery and associated gas ducts. These propulsion-related duct flows also contain high turbulence levels and high wall heat transfer rates. Current CFD codes do not provide satisfactory solutions for this general class of duct flows. Turbulence modeling is one of the significant short-comings, which is the result of inadequate physical understanding and inadequate experimental definition.

There are different types of turbulent flow such as boundary layers, shear flows from jets or wakes, and confined flows with high bulk turbulence. Each type must be analyzed and treated as a different flow phenomena. The turbulent boundary layer grows into the laminar or low-turbulence free-stream, and the turbulence properties have been successfully related to mean flow velocity gradients. Turbulent shear layers in jet mixing or wake flows are similar to the boundary layer flows in that the turbulent mixing zone propagates into the undisturbed free-stream. Turbulent duct flows are characterized by high turbulence in the bulk flow which interacts with solid boundaries and pressure gradients.

Combustion processes and combustor dilution flows produce a high turbulence field which passes through internal ducts and blade cascades. In this case, the wall viscous forces may increase or decrease the bulk turbulence intensity. This is a turbulent flow field for which the classic turbulence models are inadequate. Since the curved duct flow downstream of the turbine engine combustor contains high levels of turbulence, an improved understanding of this flow field is required to predict turbine cascade secondary flow development. Accurate predictions of secondary flows and turbulent intensity are required before heat transfer analysis techniques can be improved.

OBJECTIVES AND APPROACH

The objective of this investigation is the experimental evaluation of the influence of inlet turbulence intensity on secondary flow development in a turning duct. The existing 25.4 cm square turning duct (90°) facility, developed under contract NAS-323278, Gas Flow Environment

Nonrotating 3-D Program, is being utilized to investigate the influence of bulk turbulence levels on secondary flow development. The large scale duct flow facility allows detailed mean velocity and turbulence quantities to be measured at several streamwise planes in the curved duct. Non-intrusive laser velocimetry is being used to measure the mean and fluctuating components of velocity in all three orthogonal directions. To assure that turbulence measurements are unbiased by particle lag and other effects, comparison hot wire data will be taken to validate the L/V system calibration.

The bulk turbulence is introduced into the turning duct entrance region downstream of the inlet bell by a square bar grid (fig. 1). Maximum expected turbulence intensity behind this grid has been shown to be about 10% with a nearly uniform turbulence field (ref 1.). Additional entrance length will be added to the facility to provide sufficient mixing length down stream of the grid to provide a uniform turbulence field with a boundary layer thickness of 15% tunnel half width.

One level of turbulence intensity will be demonstrated and used in the velocity surveys of the turning duct flow field. Secondary flow development and the turbulent characteristics of the near wall flow are the primary experimental objectives. The laminar- core-flow velocity survey results from the previous contract will establish the baseline flow for assessing the effects of the increased level of turbulence intensity (ref. 2.). The turbulence and mean flow velocity results must be of sufficient quality to serve as benchmark data for CFD code development.

CURRENT STATUS AND RESULTS

The UTSI 3D LV system has been modified to improve upon the capability to reduce and analyze data in a shorter time period. This improved capability has been demonstrated and will allow increased data sample size for higher confidence in the turbulence measurements.

Additional entrance duct sections have been fabricated to provide an additional four duct widths of entrance length for a total of eight duct widths. Total pressure boundary layer surveys have confirmed a turbulent boundary layer thickness of 15–20 percent of tunnel half width at the entrance of the 90° turn. A tunnel bulk velocity of 10 m/s has been selected for the experimental conditions.

Two square-bar turbulence generator grids have been fabricated and tested for flow quality and turbulence intensity. Flow area blockages of 35 and 12 percent were evaluated for turbulence intensity generation. Hot wire data showed core levels of turbulence. two duct widths downstream of the grid. of 6 and 8 percent respectively for the two grids.

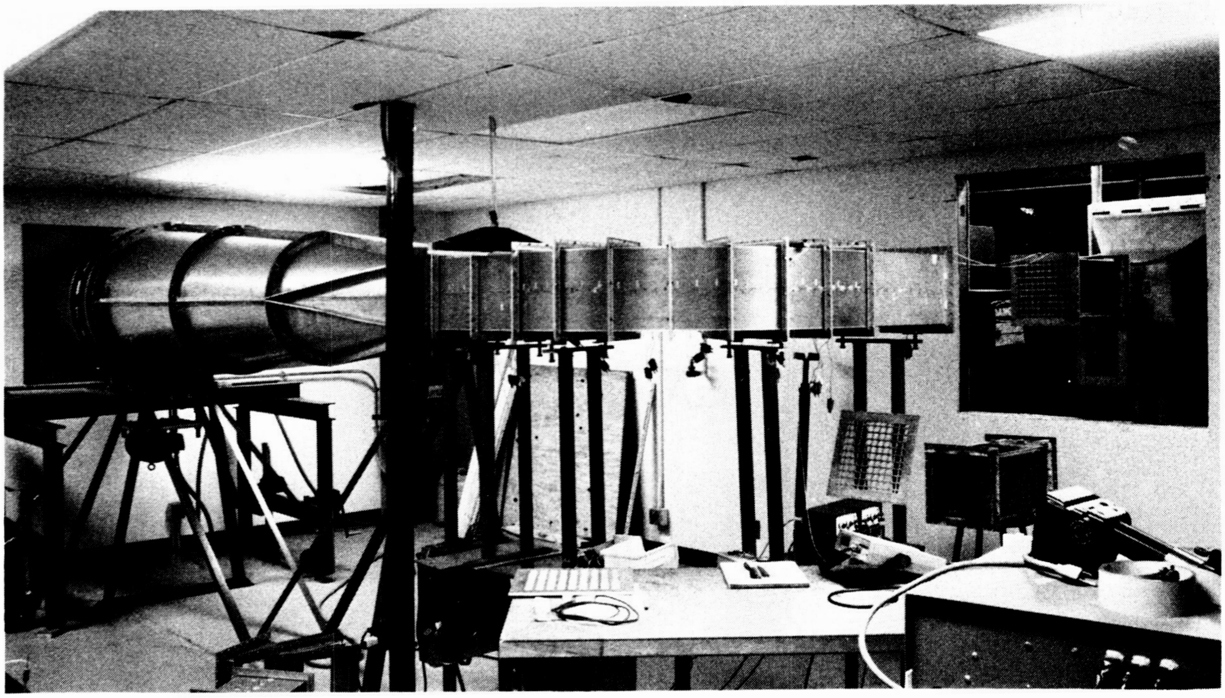
PLANS

The detailed velocity surveys with the LV system are in progress, and six duct stations will be evaluated (entrance, 0°, 30°, 60°, 90° and exit). At each station, the flow field will be measured with and without the turbulence grid for one Reynolds number corresponding to bulk velocity of 10 m/s. These two data sets, along with the thin-turbulent-boundary-layer results from the previous investigation, will allow evaluation of the effects of both inlet boundary layer thickness and bulk turbulence intensity. The resulting data bases will be documented in a form for use in CFD code development and validation.

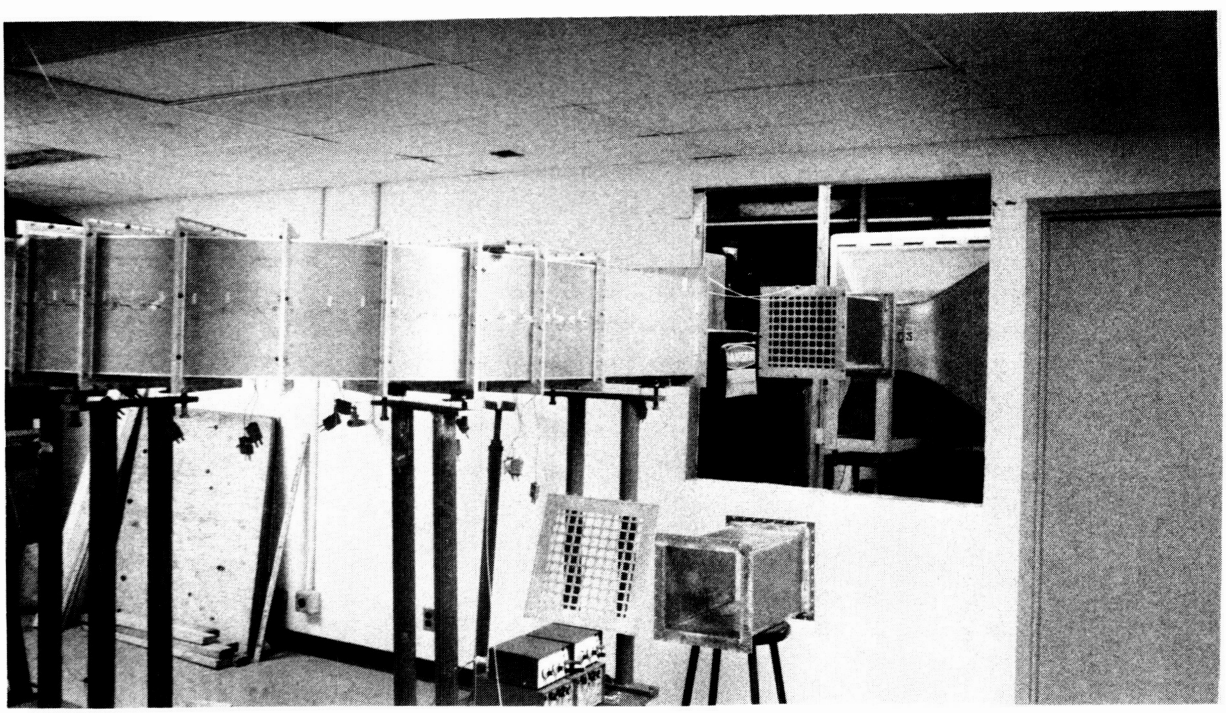
REFERENCES

1. Cenedese, A.; Iannetta, S.; Mele, P. And Morganti, M.: Turbulence Management by means of grids. Archives of Mechanics, vol. 32, 1980, pp. 633–643.

2. Crawford, R.; Peters, C.; Steinhoff, J.; Nourinejad, J.; and Ramachandran, K.: Mean Velocity and Turbulence Measurements in a 90° Curved Duct with Thin Inlet Boundary Layer. NASA CR 174811, Sept. 1985.



a. General Facility Configuration



b. Turbulence Grid Installation

Figure 1. Curved Duct Flow Facility