S1-34
186473
p. 22

N94-18556

Several Examples Where Turbulence Models Fail in Inlet Flow Field Analysis

Bernhard H. Anderson
NASA Lewis Research Center
Cleveland, OH

Inlet Flow Field Analysis
Computational Uncertainties

- Turbulence Modeling for 3D Inlet Flow Fields
  - (1) Flows Approaching Separation
  - (2) Strength of Secondary Flow Field
  - (3) 3D Flow Predictions of Vortex Liftoff
  - (4) Influence of Vortex-Boundary Layer Interactions
- Vortex Generator Modeling
  - (1) Representation of Generator Vorticity Field
  - (2) Relationship Between Generator and Vorticity Field

Inlet Flow Field Studies
Goals and Objectives

To advance the understanding, the prediction, and the control of intake distortion, and to study the basic interactions that influence this design problem.

- To develop an understanding of and predictive capability for the aerodynamic properties of intake distortion and its management.
- To establish a set of design guidelines to maximize the effectiveness of vortex flow control for the management. intake distortion

Inlet Flow Field Benchmark Data Sets
Turbulence and Vortex Generator Modeling

- Fraser Flow A, Stanford Conference 1968
- 727/JT8D-100 S-Duct Confirmation Experiment, 1973
- Univ. Tennessee Diffusing S-Duct Experiments, 1986 & 1992
- Univ. Washington TD410 Transition Duct Experiment, 1990
- M2129 Imperial College Laser-Doppler Experiment, 1990
- DRA-Bedford Experiments on the M2129 Intake S-Duct
    - (2) DRA Surface Pressure and Engine Face Experiment, 1989
    - (3) DRA Phase 1B Hot-Wire Flow Experiment, 1990
    - (4) DRA Phase 2 Yawmeter Flow Experiment, 1991
    - (5) DRA Phase 3 Vortex Generator Experiment, 1992
- TD118 Bi-Furcated Transition Duct Experiment, 1994

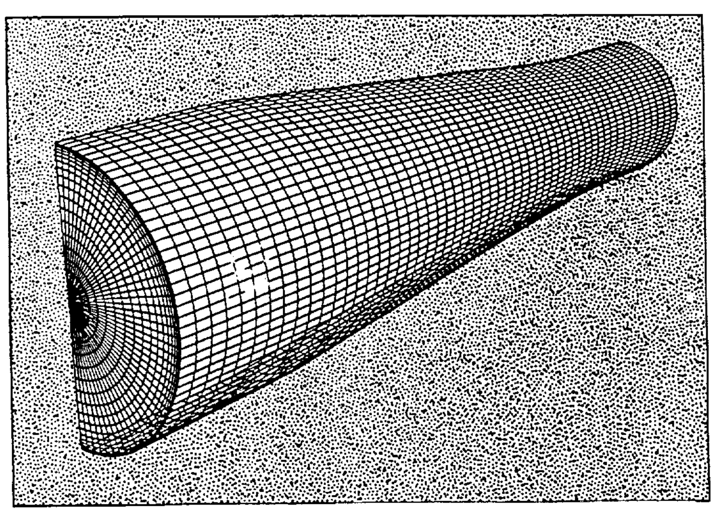
Reduced Navier-Stokes Analysis
RNS3D Computer Code

- Velocity decomposition approach, Briley and McDonald (1979 & 1984)
    - (1) Conservation form of the vorticity transport equation
    - (2) Mass flow conservation, $\dot{m} = \int_A \rho u_p dA = constant$
- Non-orthogonal coordinate system, Levy, Briley and McDonald (1983)
- Arbitrary geometry gridfile description, Anderson (1990)
    - (1) Recluster existing gridfile mesh distribution
    - (2) Redefine the centerline space curve
    - (3) Alter cross-sectional duct shape
- McDonald Camarata turbulence model

Full Navier-Stokes Analysis
PARC3D Computer Code

- Originally developed by Pulliam & Steger as AIR3D (1980)
  - (1) Conservation form of the governing equations
  - (2) Beam & Warming approximate factorization algorithm
  - (3) Central differencing within a curvilinear system
- Addition of Jameson artificial dissipation by Pulliam, ARC3D (1981)
- Developed for internal flow by Cooper, PARC3D (1987)
  - (1) Baldwin-Lomax turbulence model
  - (2) Diewert approximation to turbulence model in the reverse flow region of flow field

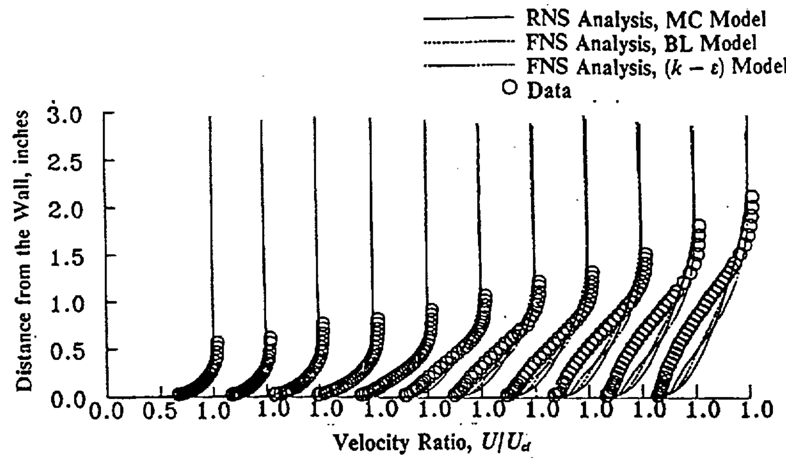
Fraser Flow A, Stanford Conference 1968
Geometry and Mesh Definition



Fraser Flow A, Stanford Conference 1968

Comparison of Turbulence Models

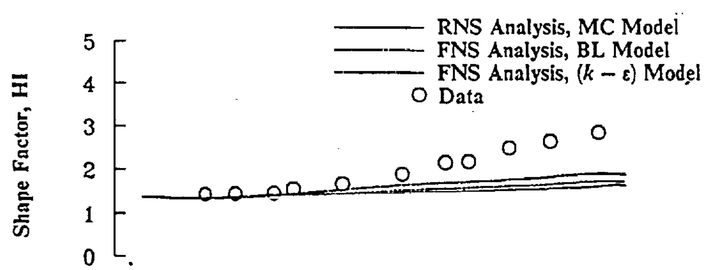
$nz = 49, y^+ = 1.17$



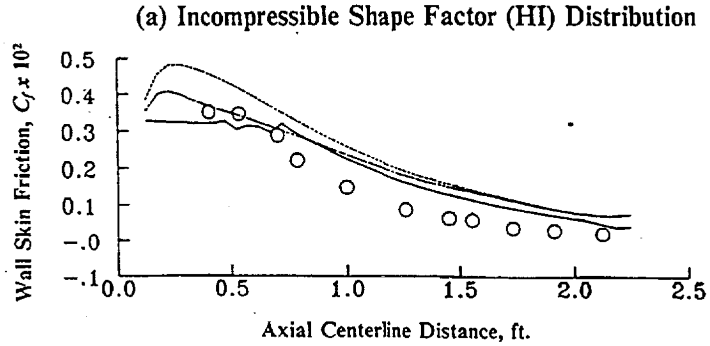
Fraser Flow A, Stanford Conference 1968

Comparison of Turbulence Models

$nz = 49, y^+ = 1.17$



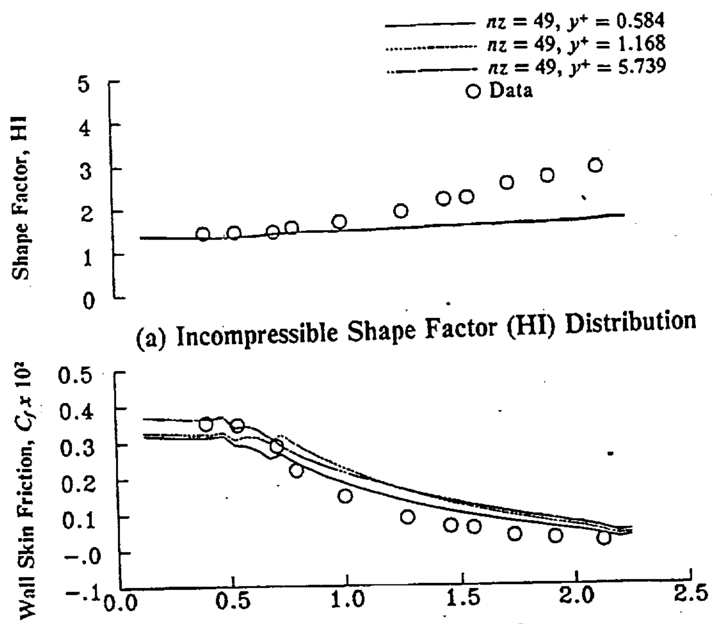
(a) Incompressible Shape Factor (HI) Distribution



(b) Wall Skin Friction (C_f) Distribution

Fraser Flow A, Stanford Conference 1968

Effect of y^+ on Flow Field Solution



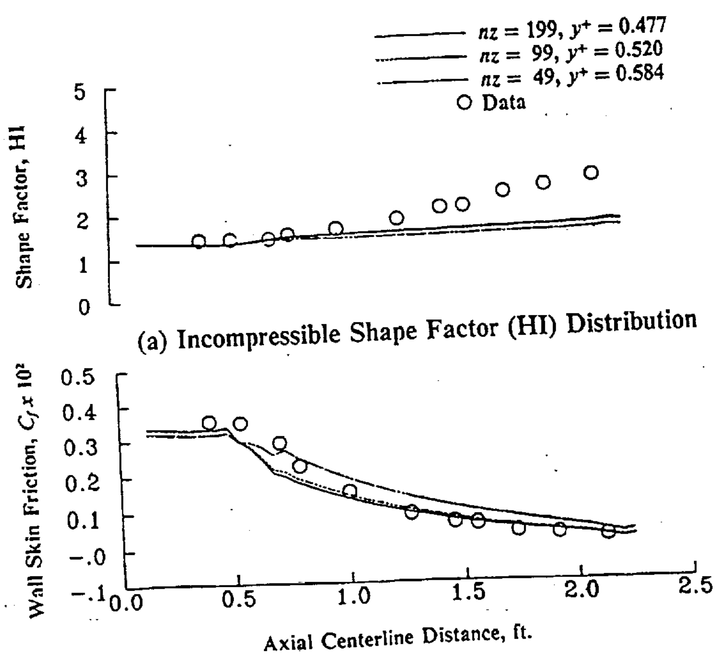
(a) Incompressible Shape Factor (HI) Distribution

(b) Wall Skin Friction (C_f) Distribution

RNS Analysis, McDonald-Camarata Model

Fraser Flow A, Stanford Conference 1968

Effect of Mesh Resolution on Flow Field Solution



(a) Incompressible Shape Factor (HI) Distribution

(b) Wall Skin Friction (C_f) Distribution

RNS Analysis, McDonald-Camarata Model

Fraser Flow A, Stanford Conference 1968

Conclusions

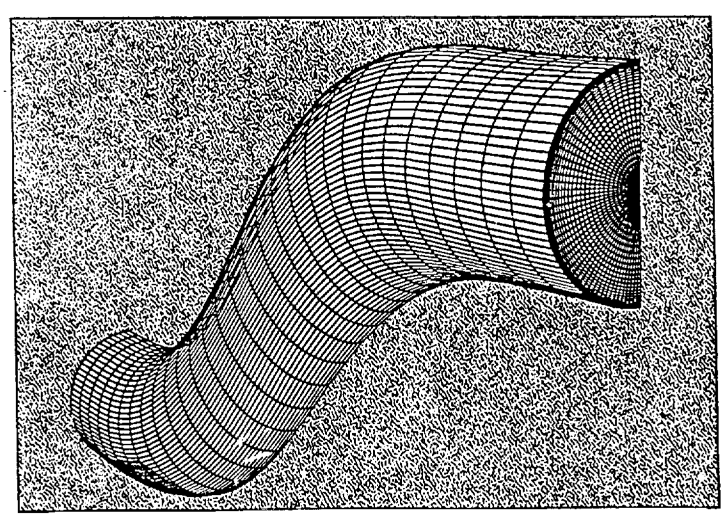
(1) The current generation of turbulence models were unable to predict the complete state of the diffuser boundary layer approaching flow separation.

(2) Both near wall and mesh resolution separately played an important role in accurate solutions to wall skin friction distribution in flows characterized as "approaching separation", but had little effect on the solution for the incompressible shape factor development.

(3) It is important that grid independent solution be demonstrated before judgements about the turbulence models be stated, and that the complete state of the wall boundary layer be considered within this evaluation.

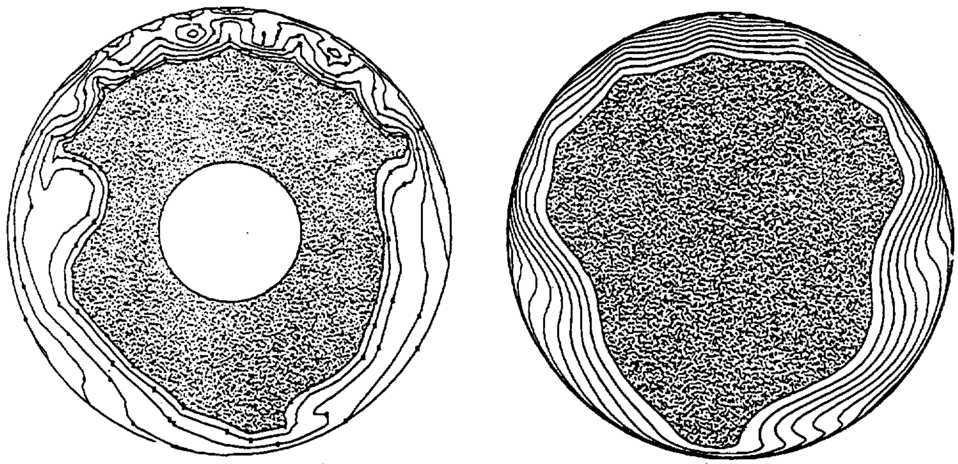
727/JT8D-100 Inlet S-Duct

Geometry and Mesh Definition



727/JT8D-100 Inlet S-Duct

Engine Face Flow Field
Generator Config. 12



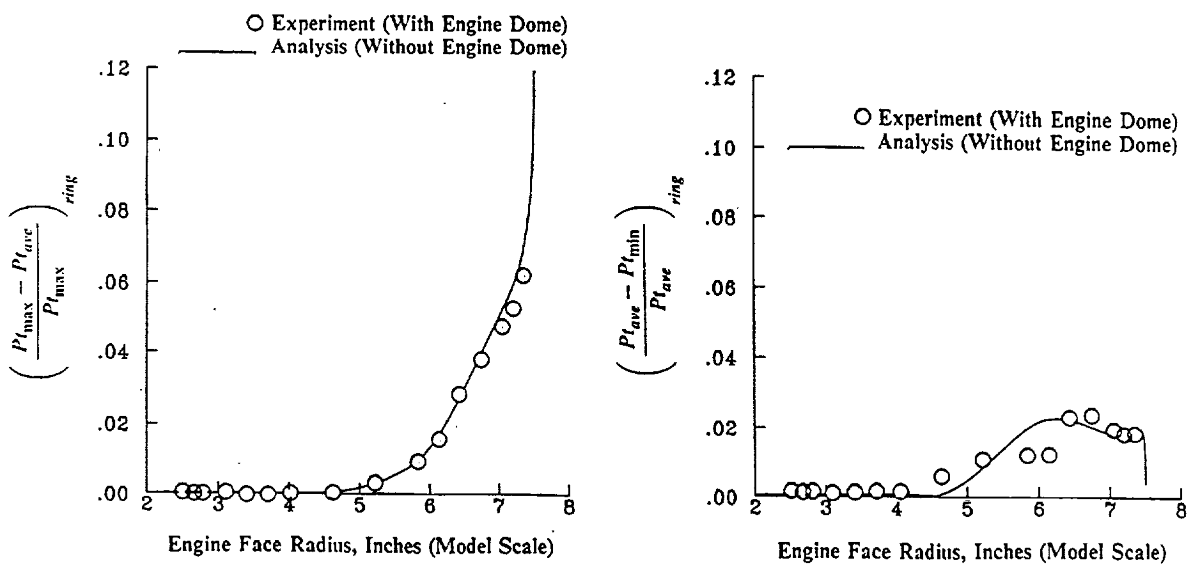
Experiment
With Engine Dome

Analysis
Without Engine Dome

RNS Analysis, McDonald-Camarata Model

727/JT8D-100 Inlet S-Duct

Engine Face Ring Distortion Characteristics
Generator Config. 12



Radial Distortion

60° Sector Circumferential Distortion

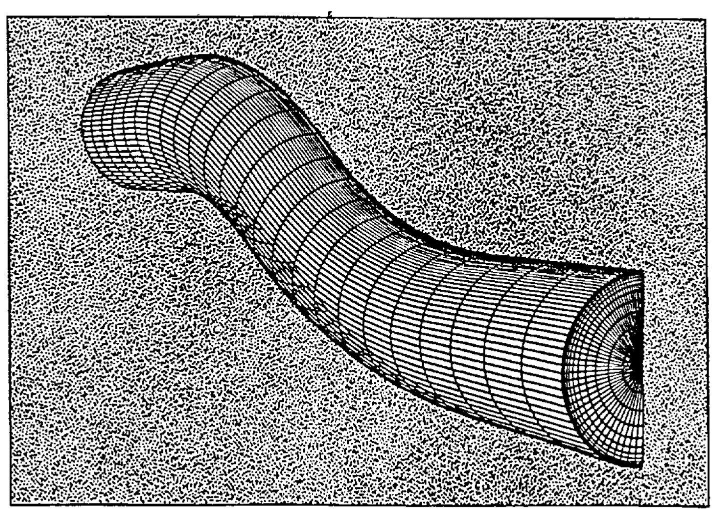
RNS Analysis, McDonald-Camarata Model

727/JT8D-100 Inlet S-Duct
Conclusions

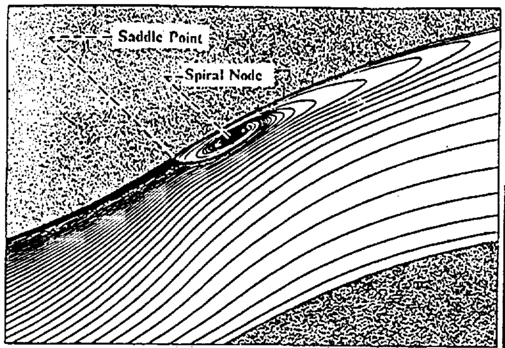
(1) The current turbulence models predict the overall performance level of vortex generator installation remarkably well, although much of the detailed flow structure was not resolved.

(2) Turbulence models in 3D inlet flow field analysis can also be evaluated on the basis of standard engine performance parameters, such as radial and circumferential ring distortion descriptors, which provide a sensitive discriminator measuring the state of the overall compressor face flow field.

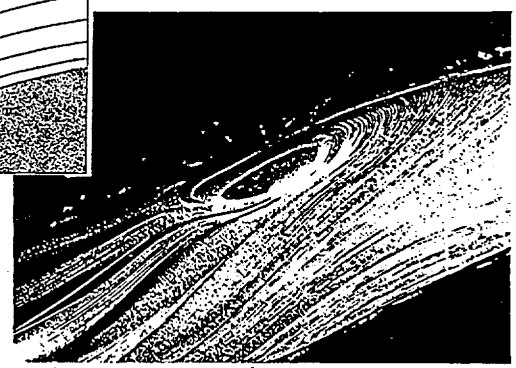
Univ. Tennessee Diffusing S-Duct
Geometry and Mesh Definition



Univ. Tennessee Diffusing S-Duct
Topology of Vortex Liftoff



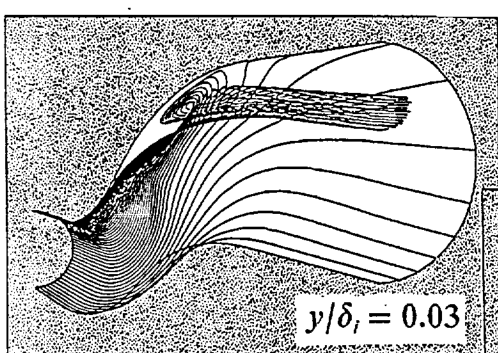
Analysis

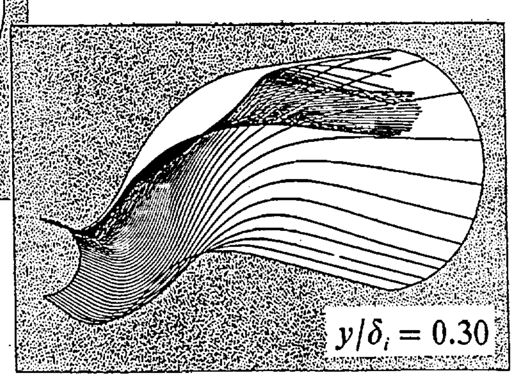


Experiment

RNS Analysis, McDonald-Camarata Model

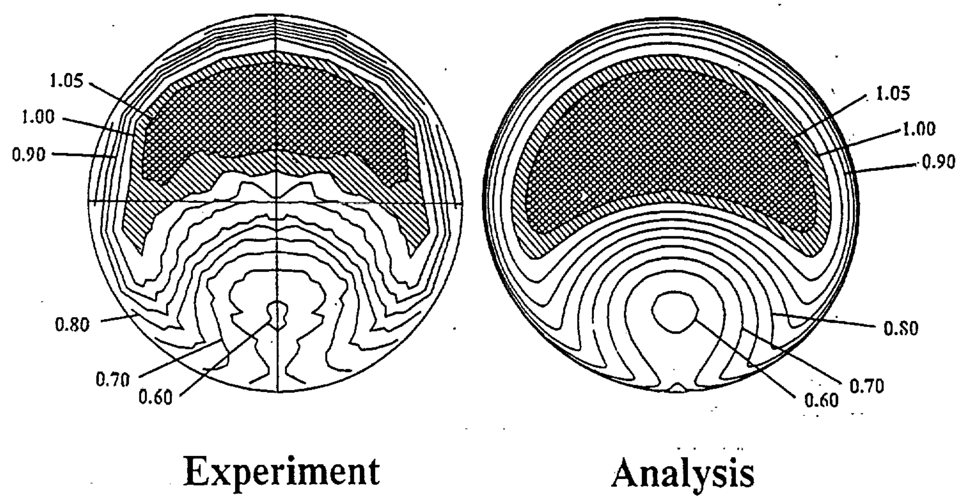
Univ. Tennessee Diffusing S-Duct
Topography of Vortex Liftoff





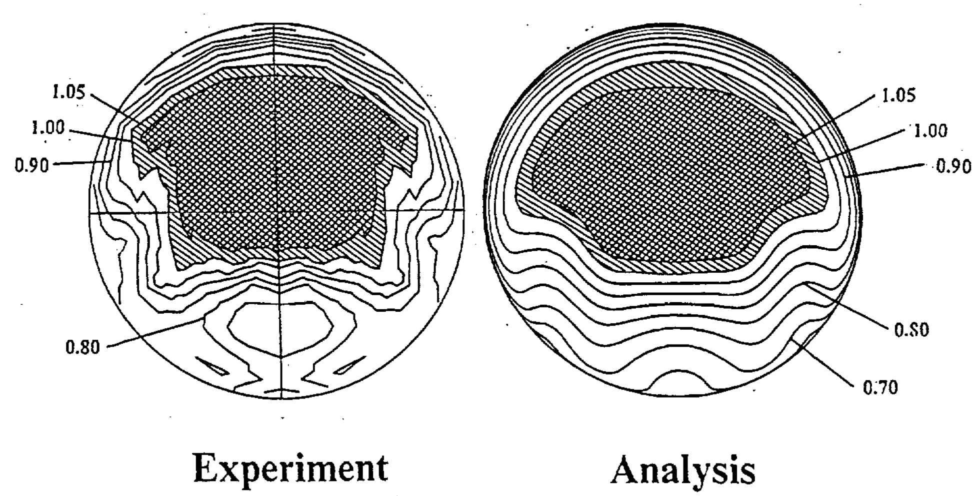
RNS Analysis, McDonald-Camarata Model

Univ. Tennessee Diffusing S-Duct
Total Pressure Coefficient Contours
Without Vortex Generators



RNS Analysis, McDonald-Camarata Model

Univ. Tennessee Diffusing S-Duct
Total Pressure Coefficient Contours
With Vortex Generators

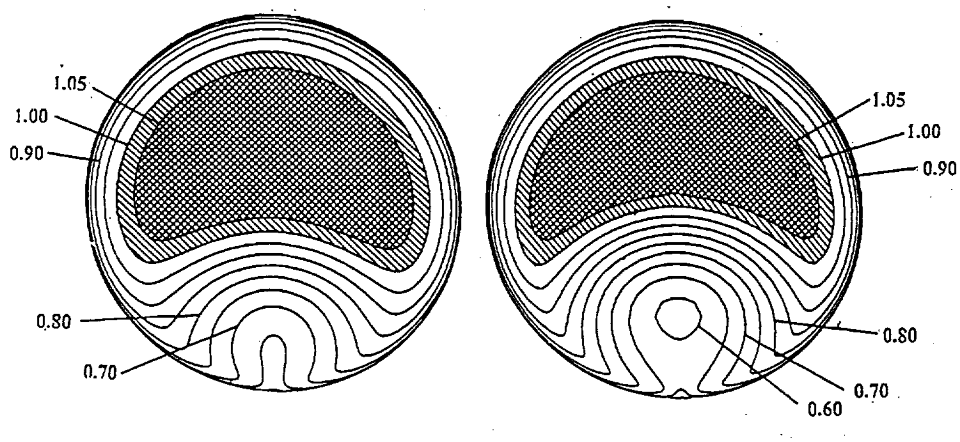


RNS Analysis, McDonald-Camarata Model

Univ. Tennessee Diffusing S-Duct

Effect of y^+ on Engine Face Flow Field

Total Pressure Coefficient Contours

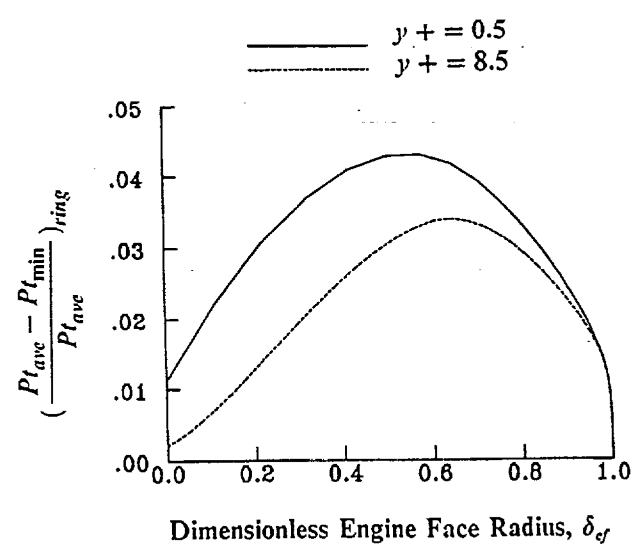


$y^+ = 8.5$ $y^+ = 0.5$

RNS Analysis. McDonald-Camarata Model

Univ. Tennessee Diffusing S-Duct

Effect of y^+ on Circumferential Distortion



RNS Analysis, McDonald-Camarata Model

Univ. Tennessee Diffusing S-Duct
Conclusions

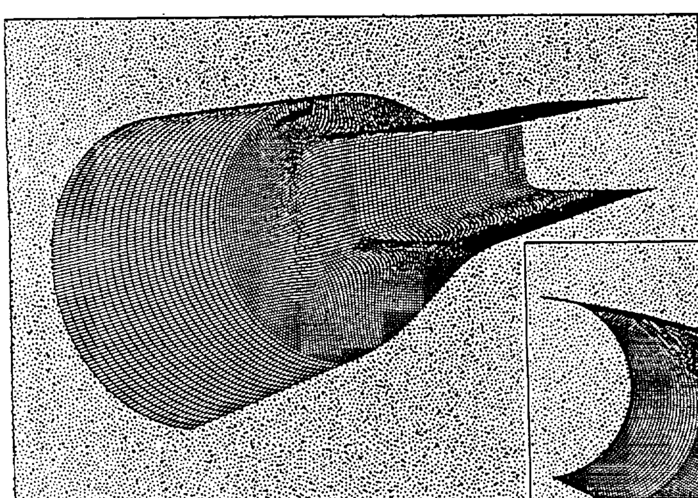
(1) Initial value space marching 3D RNS procedures adequately described the topological and topographical features of 3D flow separation associated with vortex liftoff.

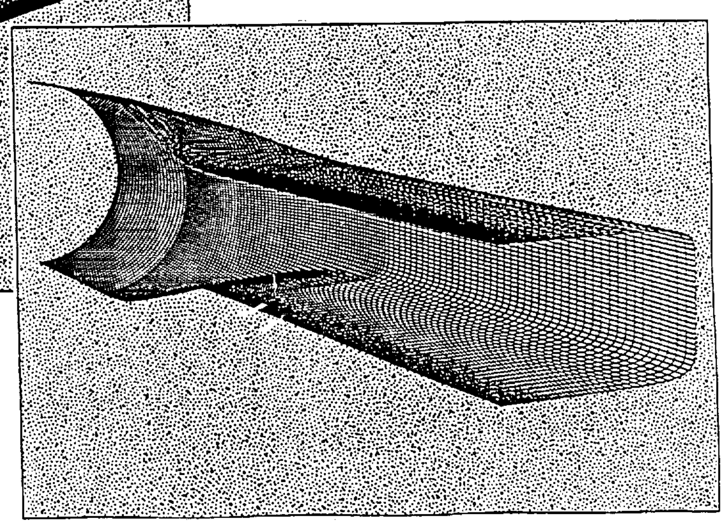
(2) The current turbulence models predicted the overall structure of vortex generator installation remarkably well, although much of the detailed flow structure was not resolved.

(3) Adequate near wall resolution was necessary to obtain an accurate solution of the phenomena of vortex lift-off.

(4) Circumferential ring distortion is a sensitive discriminator in measuring the state of the engine face flow field.

Univ. Washington TD410 Transition Duct
Geometry and Mesh Definition





Univ. Washington TD410 Transition Duct

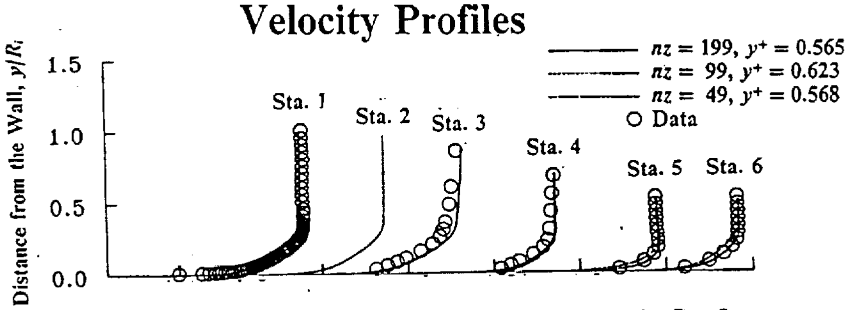
Case Definitions

Case	Grid	Total	CPU (min)	Y^+
td410.$	199 x 121 x 521	12,545,159	149.0	0.565
td410.1	99 x 121 x 521	6,241,059	73.4	0.623
td410.3	49 x 121 x 521	2,851,849	36.0	0.568
td411.1	99 x 91 x 521	4,693,689	57.1	0.623
td412.$	199 x 61 x 521	6,324,419	72.9	0.565
td412.1	99 x 61 x 521	3,146,319	38.2	0.623
td412.3	49 x 61 x 521	1,557,269	18.7	0.568
td412.4	49 x 61 x 401	1,198,589	14.7	0.568

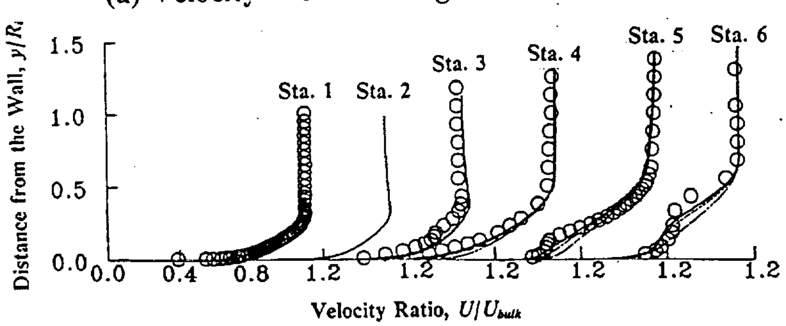
Univ. Washington TD410 Transition Duct

Effect of Radial Grid Resolution

Velocity Profiles

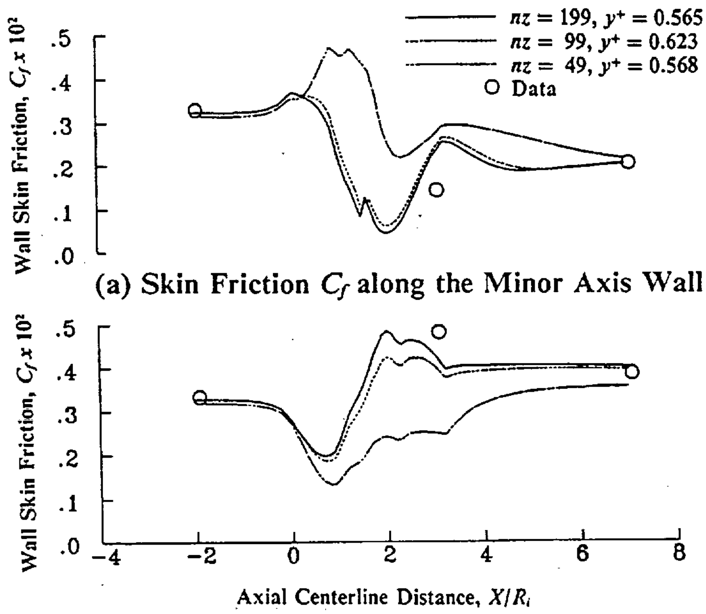


(a) Velocity Profiles Along the Minor Axis Surface



(b) Velocity Profiles Along the Major Axis Surface

Univ. Washington TD410 Transition Duct
Effect of Radial Grid Resolution
Wall Skin Friction

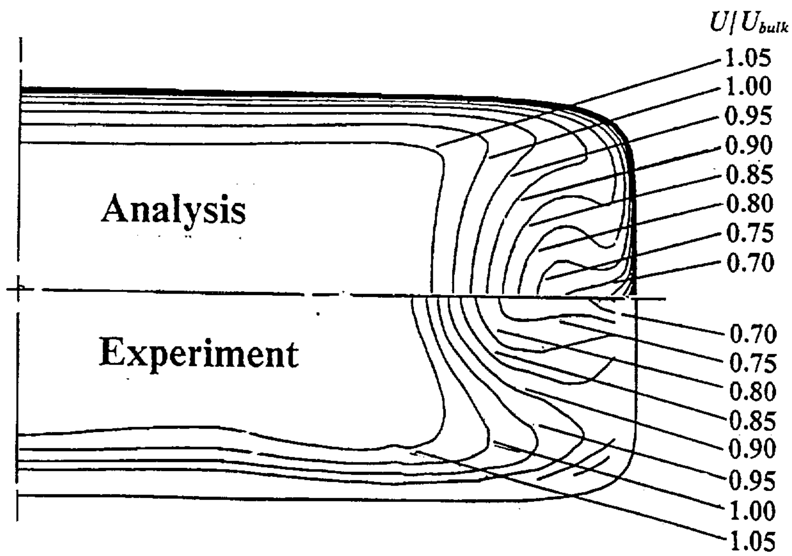


(a) Skin Friction C_f along the Minor Axis Wall

(b) Skin Friction C_f along the Major Axis Wall

RNS Analysis, McDonald-Camarata Model

Univ. Washington TD410 Transition Duct
Comparison with Experimental Data
Velocity Contours



Univ. Washington TD410 Transition Duct

Conclusions

(1) The current generation of turbulence models predicted the overall development of vortex formation reasonably well, although there were important discrepencies which could not not be explained as inadequate near wall or mesh resolution.

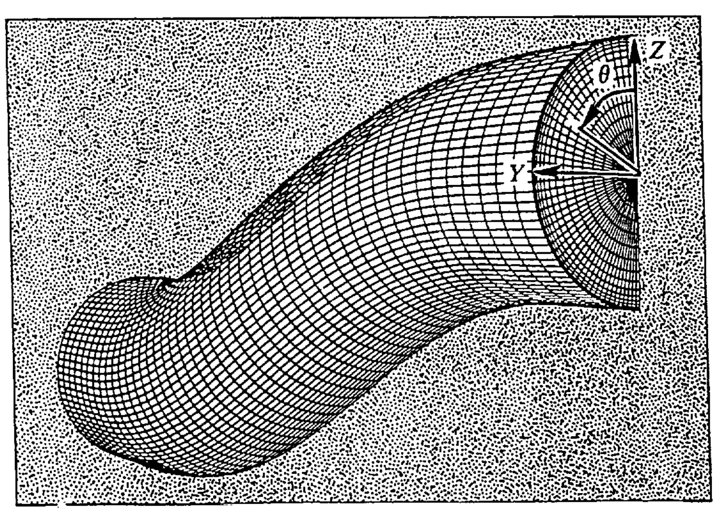
(2) Radial mesh resolution had the largest impact in the region along the major axis where the vortex pair was formed.

(3) It is important that grid independent solution be demonstrated before judgements about the turbulence models be stated.

(4) Fully 3D grid independent solutions were achieved with a Reduced Navier-Stokes analysis.

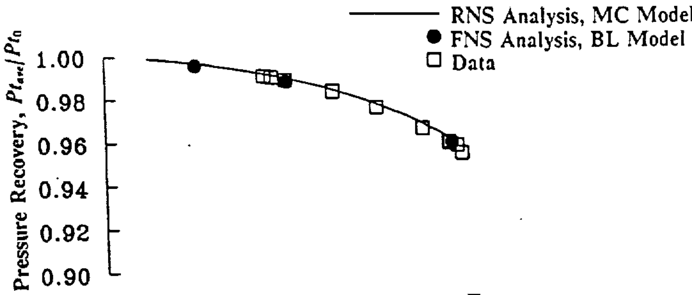
DRA M2129 Diffusing Inlet S-Duct

Geometry and Mesh Definition

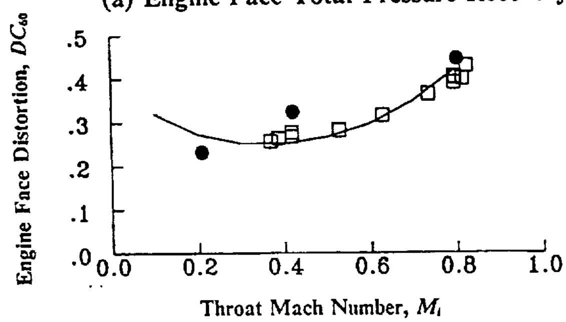


DRA M2129 Diffusing Inlet S-Duct

Performance Characteristics



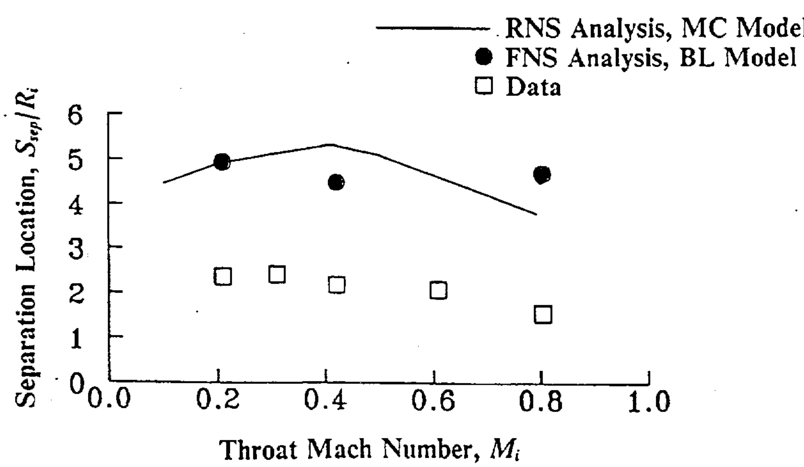
(a) Engine Face Total Pressure Recovery



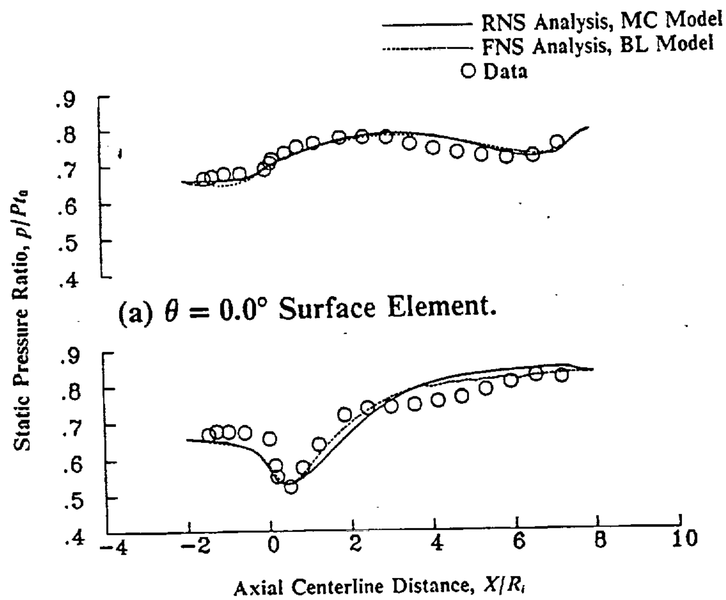
(b) Engine Face DC_{60} Distortion

DRA M2129 Diffusing Inlet S-Duct

Separation Characteristics



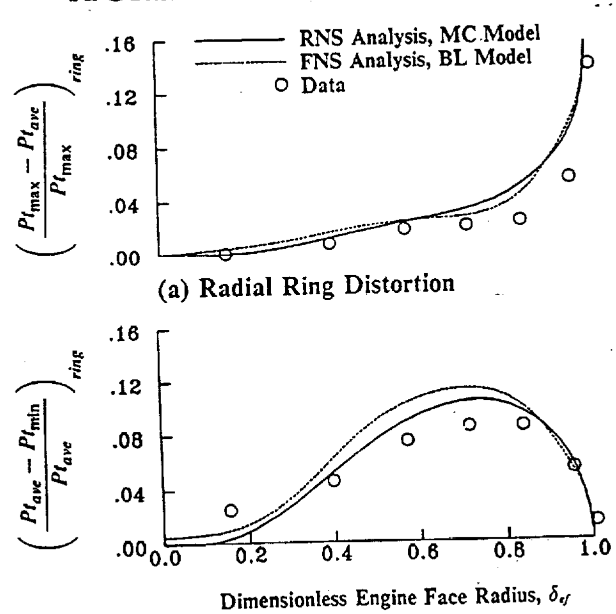
DRA M2129 Inlet S-Duct
Wall Static Pressure Distribution
AGARD Test Case 3.1



(a) $\theta = 0.0°$ Surface Element.

(b) $\theta = 180.0°$ Surface Element.

DRA M2129 Diffusing Inlet S-Duct
Engine Face Distortion Characteristics
AGARD Test Case 3.1



(a) Radial Ring Distortion

(b) 60°-Sector Circumferential Distortion

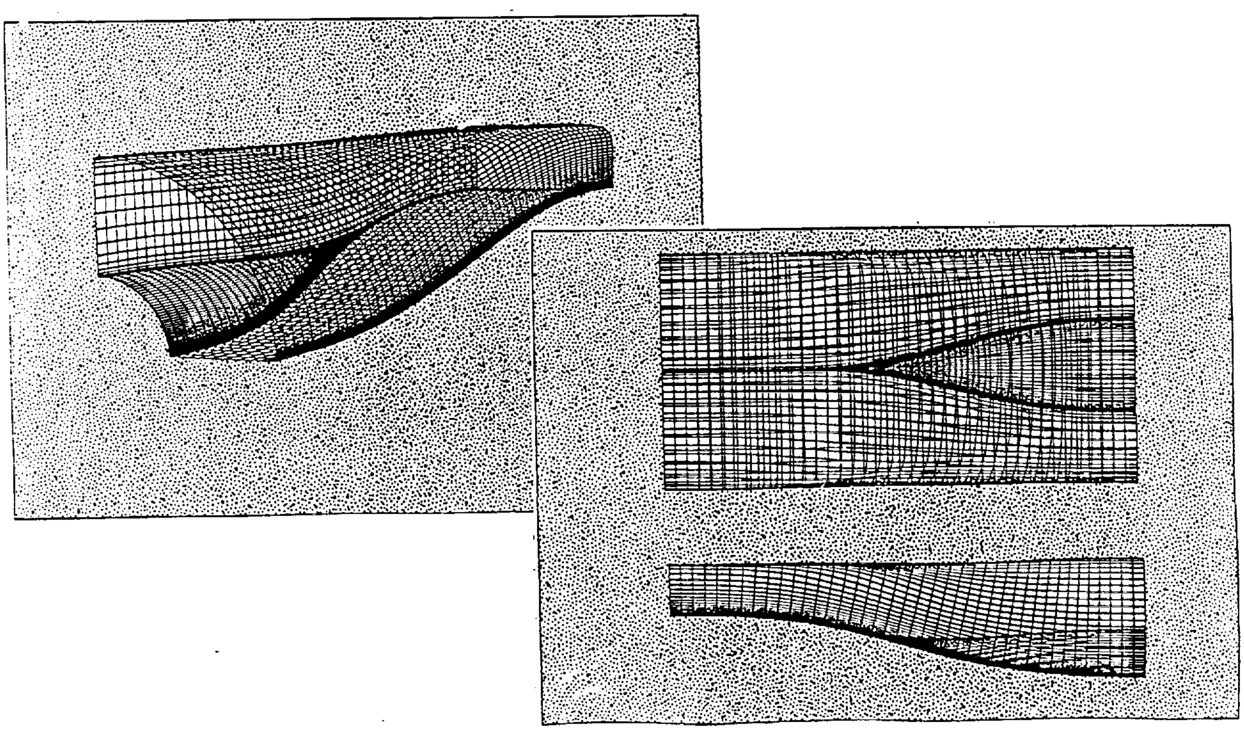
DRA M2129 Inlet S-Duct
Conclusions

(1) Both Full Navier-Stokes (FNS) and Reduce Navier-Stokes (RNS) analyses adequately describe the overall flow physics of vortex liftoff, but consistently predict the location of liftoff further downstream in the duct inlet than was indicated by data.

(2) The current generation of turbulence models were unable to desribed the influence of separation on the main pressure field for "strong" vortex liftoff interactions.

(3) The current generation of mixing length turbulence models give remarkable good performance results, while the existing discrepancies between data and analysis can be attributed primarily to the over prediction of the liftoff location.

TD118 Bi-Furcated Transition Duct
Geometry and Mesh Definition



TD118 Bi-Furcated Transition S-Duct
Research Objectives

- To demonstrate diffuser duct technology advancement by using CFD to design a "conventionally shorter" transitioning S-duct configuration for application towards high speed inlet systems.
- To develop a computational protocol whereby turbulence model evaluations can be made between different computer codes.
- To develop a benchmark data set to evaluate CFD analysis and turbulence models, which cover fundamental flow phenomena as well as overall flow field physics as determined by standard engine performance parameters.

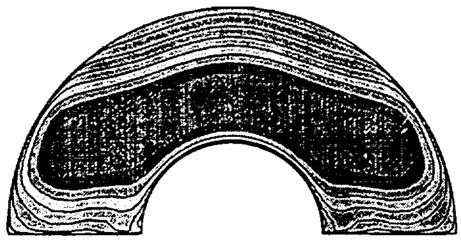
TD118 Bi-Furcated Transition S-Duct
Effect of Turbulence Model on Inlet Performance

Analysis	Turbulence Model	Pt_{ave}/Pt_0	DH	DC_{60}
RNS3D	McDonald-Camarata	0.955	0.167	0.116
PARC3D	Baldwin-Lomax	0.959	0.163	0.135
PARC3D	P.D. Thomas	0.970	0.153	0.073
PARC3D	Launder-Spaulding	0.975	0.150	0.177

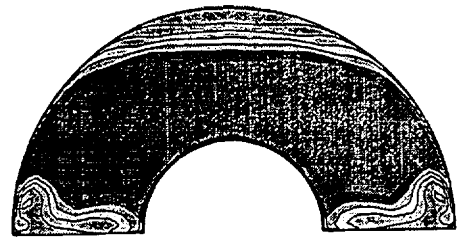
$$DH = (Pt_{max} - Pt_{min})/Pt_{ave}$$

$$DC_{60} = (Pt_{ave} - Pt_{min,60})/q_{ave}$$

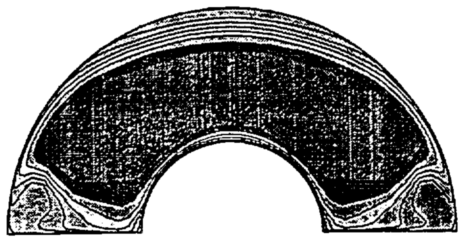
TD118 Bi-Furcated Transition S-Duct
Effect of Turbulence Model on Engine Face Flow Field



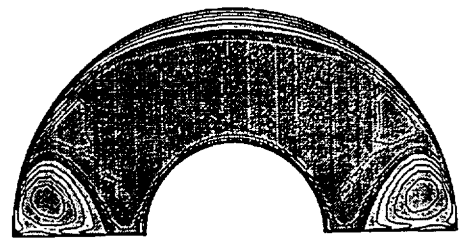
(a) RNS Analysis, McDonald-Camarata Model



(c) FNS Analysis, P. D. Thomas Model

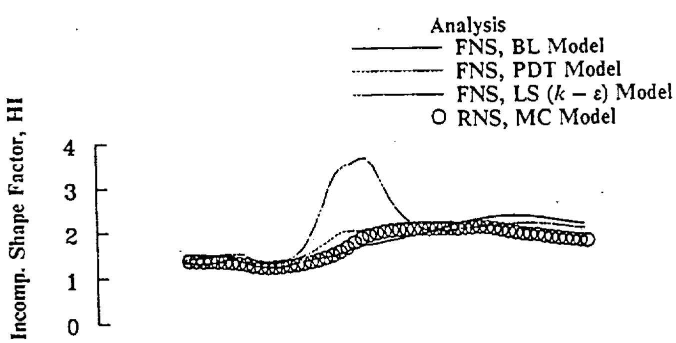


(b) FNS Analysis, Baldwin-Lomax Model

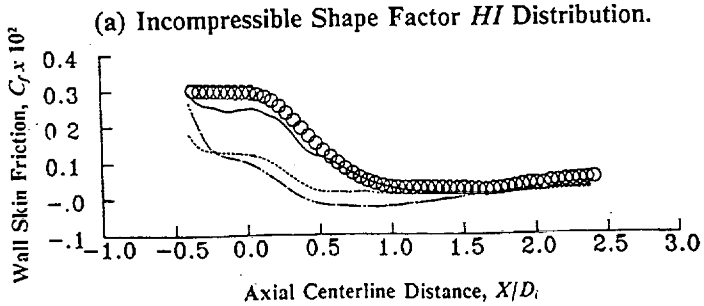


(d) FNS Analysis, Launder-Spalding ($k - \varepsilon$) Model

TD118 Bi-Furcated Transition S-Duct
Effect of Turbulence Model on Wall Boundary Layer
$\theta = 90.0°$ Surface Element

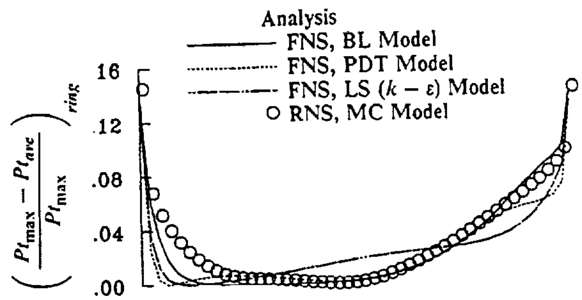


(a) Incompressible Shape Factor HI Distribution.

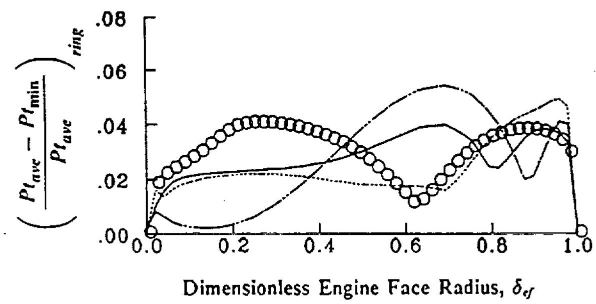


(b) Wall Skin Friction C_f Distribution.

TD118 Bi-Furcated Transition Duct
Effect of Turbulence Model on Ring Distortion



(a) Radial Ring Distortion



(b) 60°-Sector Circumferential Distortion

Inlet Flow Field Analysis
Concluding Remarks

(1) Difficulties in complex 3D flow fields often arise because fundamental 2D aerodynamic interactions have not been adequately resolved.

(2) Near wall (y^+) and radial mesh resolution (nz) play an important role in fundamental 2D and complex 3D flow field analysis.

(3) Judgements about turbulent models should not be stated until grid independent solutions have been established.

(4) Adequateness of turbulence models in inlet flow field analysis should also be made on the basis of fundamental performance parameters used to quantify the "goodness" of the flow entering the engine.