A SUMMARY OF COMPUTATIONAL EXPERIENCE AT GE AIRCRAFT ENGINES FOR COMPLEX TURBULENT FLOWS IN GAS TURBINES

R. Zerkle and C. Prakash
GE Aircraft Engines
Cincinnati, Ohio

CONTENTS:

INTRODUCTION:

- Indications are that the standard $k-\varepsilon$ turbulence model together with standard wall functions are adequate for CFD simulations in cavities away from the primary gaspath of a gas turbine engine.
- However, CFD simulations in the primary gaspath and in blade cooling passages require more advanced turbulence models.
- Therefore, this presentation will summarize some CFD experience at GEAE only for flows in the primary gaspath of a gas turbine engine and in turbine blade cooling passages.

2-D BOUNDARY LAYER CODE WITH LOW REYNOLDS NUMBER (LRN) TURBULENCE MODEL:

- The STAN5 B.L. code was modified to include the LRN $k-\varepsilon$ turbulence model of Lam & Bremhorst as described by Zerkle & Lounsbury [1].
- Includes the following near-wall effects:
  - High freestream turbulence
  - Axial pressure gradient
  - Onset of transition
  - Relaminarization
  - Wall roughness
  - Wall curvature
- Used to compute heat transfer coefficient distributions on turbine airfoil external surfaces.
- Primary limitation:
  - It's a 2-D code in a 3-D environment.

3-D NAVIER-STOKES CODE WITH WALL FUNCTIONS:

- Time-marching finite-volume formulation of the Reynolds-averaged Navier-Stokes equations as described by Turner & Jennions [2,3].
- Includes:
  - Explicit Runga-Kutta flow solver
  - Implicit formulation of the standard $k-\varepsilon$ turbulence model
  - Standard wall functions
  - Transonic flow effects
- Used to simulate high speed flows in turbomachinery passages.
- Limitations:
  - Lacks near-wall physics of the 2-D boundary layer code.
  - For example, lack of boundary layer transition leads to overprediction of loss for some turbomachinery airfoil passages containing significant regions of transitional flow.

3–D NAVIER–STOKES CODE WITH LOW REYNOLDS NUMBER (LRN) TURBULENCE MODEL:

- The LRN $k-\varepsilon$ turbulence model of Lam & Bremhorst was implementec in the 3–D Navier–Stokes code as described by Dailey, Jennions anc Orkwis [4].
- Addition of the LRN turbulence model improved the prediction of los: for transitional flows.
- Primary limitation:
  - The need for a very fine grid in the near–wall region leads to exces sive run times which renders the code impractical for design applica tions at this time.

FILM COOLING SIMULATION:

- Film cooling at the surface of an HP turbine airfoil is crucial to its life.
- Improvement of the film cooling process would significantly improve turbine performance by reducing the need for cooling air flow.
- CFD simulation could facilitate film cooling development by reducing the need for expensive cascade testing and, more importantly, by giving greater insight into the associated flow physics.
- A CFD simulation of film–cooling tests, which were carried out at the Univ. of Texas by Professors Crawford & Bogard, and their students, is described by Leylek & Zerkle [5].
- These tests are of special interest because the ranges of film coolinc parameters are consistent with those typically found in gas turbine airfoil applications.
- The objective was to validate a CFD model of film cooling by comparinc numerical and experimental results.

FILM COOLING SIMULATION (CONT'D):

- The model includes:
  - A 3-D, fully-elliptic, Navier-Stokes solution of the coupled flow in the plenum, film hole, and cross-stream regions.
  - An exact representation of the inclined, round, film-hole geometry using a highly-orthogonalized fine grid mesh.
  - The standard k-ε turbulence model with standard wall functions.

FILM COOLING SIMULATION (CONT'D):

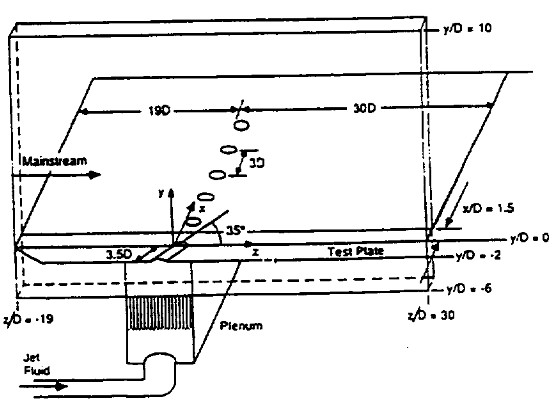


Figure 1. Essential features of experimental film cooling configuration showing overall extent of computation domain and coordinate system.

FILM COOLING SIMULATION (CONT'D):

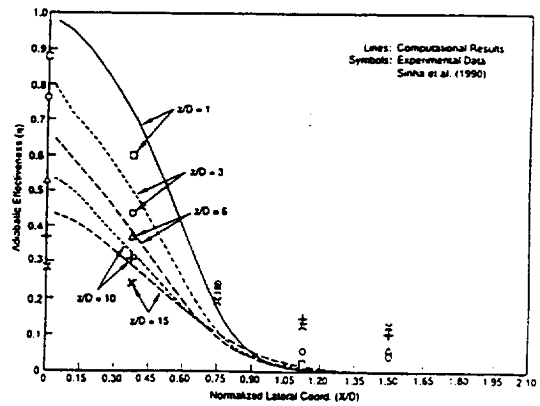


Figure 14. Lateral variation of adiabatic effectiveness from computations and experiments for M=0.5 at five streamwise stations.

FILM COOLING SIMULATION (CONT'D):

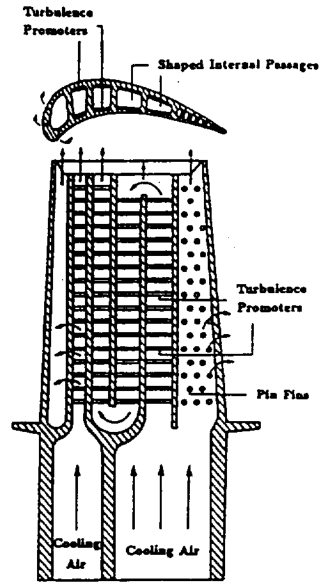
- Summary of Results:
  - The flowfield is dominated by a strong three–way coupling between the plenum, film–hole, and cross–stream regions.
  - Flow within the film hole is extremely complex, with counter–rotating vortices and local jetting effects.
  - A comparison of computed and experimental film effectiveness on the plate surface indicates that the simulated coolant jet is not spreading as fast as experimental results.
- Conclusions:
  - There is excellent *qualitative* agreement between the numerical and experimental results.
  - However, the lack of lateral spreading of the coolant is caused by the inability of the k–ε turbulence model to cope with non–uniform rates of diffusion in different directions.
  - Improved accuracy requires an *anisotropic* turbulence model.

TURBULATED PASSAGE SIMULATION:

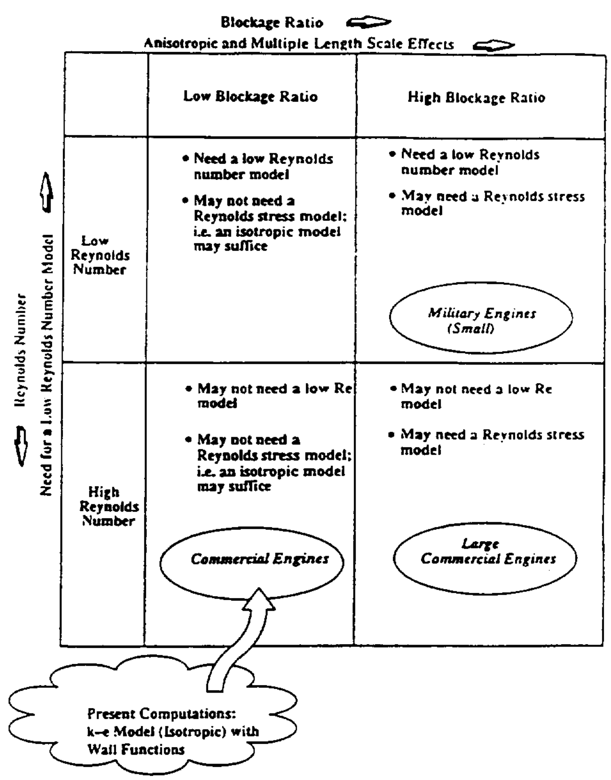
- Modern high–performance turbine blades are cooled by internal radially–rotating serpentine passages.
- The air flowing through these passages is exposed to very large Coriolis and centripetal body forces which induce strong secondary flows and buoyant effects.
- These effects tend to increase heat transfer coefficient on the trailing face of an up–pass, but decrease it on the leading face.
- Turbulators are added to the passage walls in order to enhance their cooling effectiveness.
- The primary objective of blade cooling development is to determine turbulator and passage configurations which can influence the secondary flows to achieve a uniformily high heat transfer coefficient, but within pressure–drop constraints.
- Rotating–passage rig tests are expensive, and it is very difficult to achieve high–quality data in the range of engine operating parameters.

TURBULATED PASSAGE SIMULATION (CONT'D):

- Therefore, CFD could facilitate blade cooling development by simulating new cooling configurations at real engine operating conditions.
- An exploratory investigation of CFD simulation in turbulated blade cooling passages is described by Prakash & Zerkle [6].
- Conclusions are:
  - The flow fields in turbulated blade cooling passages are very complex, and desired accuracy requires advanced turbulence models.
  - An LRN model is needed near turbulated walls in the case of low passage Reynolds number.
  - An anisotropic turbulent model is needed in the case of large blockage ratio (rib height to passage diameter).
  - Practical LRN and anisotropic models are not yet available.



Cooling concepts of a modern multipass turbine blade



OVERALL CONCLUSIONS:

- Application of the standard k−ε turbulence model with wall function is not adequate for accurate CFD simulation of aerodynamic perfor mance and heat transfer in the primary gas path of a gas turbine engine
- New models are required in the near−wall region which include mor physics than wall functions. The two−layer modeling approach ap pears attractive because of its computational economy.
- In addition, improved CFD simulation of film cooling and turbine blad internal cooling passages will require anisotropic turbulence models
- New turbulence models must be practical in order to have a significar impact on the engine design process.
- A coordinated turbulence modeling effort between NASA center would be beneficial to the gas turbine industry.

LIST OF REFERENCES:

1. Zerkle, R. D., and Lounsbury, R. J., 1989, "Freestream Turbulence Effect on Turbin Airfoil Heat Transfer," AIAA *Journal of Propulsion*, Vol. 5, pp. 82−88.

2. Turner, M. G., and Jennions, I. K., 1993, "An Investigation of Turbulence Modelin in Transonic Fans Including a Novel Implementation of an Implicit k−ε Turbulence Mod el," ASME *Journal of Turbomachinery*, Vol. 115, pp. 249−260.

3. Jennions, I. K., and Turner, M. G., 1993, "3−D Navier−Stokes Computations o Transonic Fan Flow Using an Explicit Flow Solver and an Implicit k−ε Solver," ASMI *Journal of Turbomachinery*, Vol. 115, pp. 261−272.

4. Dailey, L. D., Jennions, I. K., and Orkwis, P. D., 1994, "Simulating Laminar−Turbu lent Transition With a Low Reynolds Number k−ε Turbulence Model in a Navier−Stoke Flow Solver," AIAA Paper 94−0189.

5. Leylek, J. H., and Zerkle, R. D., 1994, "Discrete−Jet Film Cooling: A Compariso of Computational Results With Experiments," ASME *Journal of Turbomachinery*, Vo 116, pp. 358−368.

6. Prakash, C., and Zerkle, R. D., 1993, "Prediction of Turbulent Flow and Heat Trans fer in a Ribbed Rectangular Duct With and Without Rotation," ASME Paper 93−GT−20