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 & Lounsbery [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 Runge-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 implemented in the 3-D Navier-Stokes code as described by Dailey, Jennions and Orkwis [4].

- Addition of the LRN turbulence model improved the prediction of loss for transitional flows.

- Primary limitation:
  - The need for a very fine grid in the near-wall region leads to excessive run times which renders the code impractical for design applications 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 cooling parameters are consistent with those typically found in gas turbine airfoil applications.

- The objective was to validate a CFD model of film cooling by comparing 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 $k-\varepsilon$ turbulence model with standard wall functions.

Figure 1. Essential features of experimental film cooling configuration showing overall extent of computation domain and coordinate system.
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 $k-\varepsilon$ 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 uniformly 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.
### Turbulence Promoter

- Shaped Internal Passages

### Cooling Concepts of a Modern Multipass Turbine Blade

#### Blockage Ratio
- **Anisotropic and Multiple Length Scale Effects**

<table>
<thead>
<tr>
<th>Blockage Ratio</th>
<th>Low Blockage Ratio</th>
<th>High Blockage Ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Low Blockage Ratio</strong></td>
<td>- Need a low Reynolds number model&lt;br&gt;- May not need a Reynolds stress model, i.e., an isotropic model may suffice</td>
<td>- Need a low Reynolds number model&lt;br&gt;- May need a Reynolds stress model</td>
</tr>
<tr>
<td><strong>High Blockage Ratio</strong></td>
<td>- May not need a low Reynolds number model&lt;br&gt;- May not need a Reynolds stress model, i.e., an isotropic model may suffice</td>
<td>- May not need a low Reynolds number model&lt;br&gt;- May not need a Reynolds stress model</td>
</tr>
</tbody>
</table>

- **Present Computations:** k-ω Model (Anisotropic) with Wall Functions

- **Military Engines** (Small)
- **Commercial Engines**
- **Large Commercial Engines**
OVERALL CONCLUSIONS:

- Application of the standard k-ε turbulence model with wall function is not adequate for accurate CFD simulation of aerodynamic performance and heat transfer in the primary gas path of a gas turbine engine.

- New models are required in the near-wall region which include more physics than wall functions. The two-layer modeling approach appears attractive because of its computational economy.

- In addition, improved CFD simulation of film cooling and turbine blade internal cooling passages will require anisotropic turbulence models.

- New turbulence models must be practical in order to have a significant impact on the engine design process.

- A coordinated turbulence modeling effort between NASA centers would be beneficial to the gas turbine industry.

LIST OF REFERENCES:


