JOURNAL OF PROPULSION AND POWER
Vol. 11, No. 2, March–April 1995

Chimera Grids in Computing Flowfields in Turbine-Blade-Internal-Coolant Passages

M. A. Stephens,* M. J. Rimlinger,* and T. I-P. Shih†
Carnegie Mellon University, Pittsburgh, Pennsylvania 15213
and
K. C. Civinskas‡
NASA Lewis Research Center, Cleveland, Ohio 44135

When computing flows inside geometrically complex turbine-blade coolant passages, the structure of the grid system used can significantly affect the overall time and cost required to obtain solutions. This article addresses this issue while evaluating and developing computational tools for the design and analysis of coolant-passages, and is divided into two parts. In the first part, the various types of structured and unstructured grids are compared in relation to their ability to provide solutions in a timely and cost-effective manner. This comparison shows that the overlapping structured grids, known as chimera grids, can rival and in some instances exceed the cost-effectiveness of unstructured grids in terms of both the man hours needed to generate grids and the amount of computer memory and CPU time needed to obtain solutions. In the second part, a computational tool utilizing chimera grids was used to compute the flow and heat transfer in two different turbine-blade coolant passages that contain baffles and numerous pin fins. These computations showed the versatility and flexibility offered by chimera grids.

Introduction

TO increase thermal efficiency and specific thrust, advanced gas turbine engines operate at very high turbine inlet temperatures. In order to accommodate such high inlet temperatures, the turbine blades must be cooled so that their structural integrity can be maintained. One effective way of cooling the turbine blades is to introduce coolant air from the compressor into the interior of the blades.[1–6] Typically, the coolant air enters each blade from the hub section and exits through the blade's trailing edge. While inside the blade, coolant air flows through a complicated passage that may be serpentine and contains a number of channels, pin fins, and transverse ribs.

The enormous complexity of the geometry coupled with rotation and turbulence make the flow within coolant passages extremely complicated. In order to design "effective" coolant passages, it is essential that there be a good understanding of 1) the multidimensional flow physics taking place within coolant passages and 2) how design and operating parameters affect the flowfield. The understanding that is needed can be obtained by experimental or mathematical methods. The major objective of this investigation is to evaluate and develop mathematical approaches, or more specifically computational tools, that can potentially be used to obtain the required understanding and be able to aid in the design and analysis of coolant passages.

Presented as Paper 93-2559 at the AIAA/SAE/ASME/ASEE 29th Joint Propulsion Conference and Exhibit, Monterey, CA, June 28–30, 1993; received July 21, 1993; revision received May 6, 1994; accepted for publication June 14, 1994.

*Graduate Student, Department of Mechanical Engineering. Student Member AIAA.

†Professor, Department of Mechanical Engineering. Member AIAA.

‡Deputy Branch Chief, Turbomachinery Technology Branch. Member AIAA.

In order for a computational tool to be useful in the design and analysis of coolant passages, it must be able to generate physically correct solutions, and do so in a timely and cost-effective manner. By timely and cost-effectiveness, it is meant not just the computer time and cost. Instead, it is meant the total amount of time and cost needed to generate a solution, including the time and cost needed to setup the problem before it can be submitted to the computer (e.g., grid generation, boundary, and initial conditions, etc.), as well as the computer time and cost. The focus of this investigation is on ways to obtain solutions that are timely and cost-effective. Readers interested in accuracy or correctness of solutions are referred to Prakash and Zerkle[7] and references cited therein for errors associated with modeling of physics (e.g., turbulence); and Steinthorsson et al.[8] for errors associated with "numerics."

There are several factors that affect how timely and cost-effective solutions can be obtained for the multidimensional flowfield within realistic coolant passages. Of these, the most important factor is associated with the amount of time and effort needed to generate the grid that represents the geometry of the coolant passage. This is especially true for complex-shaped coolant passages such as those in radial turbine blades that have highly twisted flow passages with changing cross sections, baffles, and numerous pin fins (e.g., see Fig. 1). For such coolant-passage geometries, depending upon the structure of the grid, a grid can take up to one man-month or more to generate.[9,10] If many different coolant-passage designs need to be evaluated, then the amount of man-months required is compounded since many different grids must be generated.

At this point, it is noted that despite the effort-intensive nature of grid generation, no shortcuts can be made in it. There are two reasons for this. First, grid generation produces the mathematical representation of the problem's geometry; and, in fluid mechanics, geometry has first-order effects on the nature of the flow. Second, the structure of the grid (i.e., how grid points or cells are connected to each other) affects significantly the efficiency with which solutions can be obtained on the computer.

The organization of the rest of this article is as follows: First, the different types of grid structures that can be em-

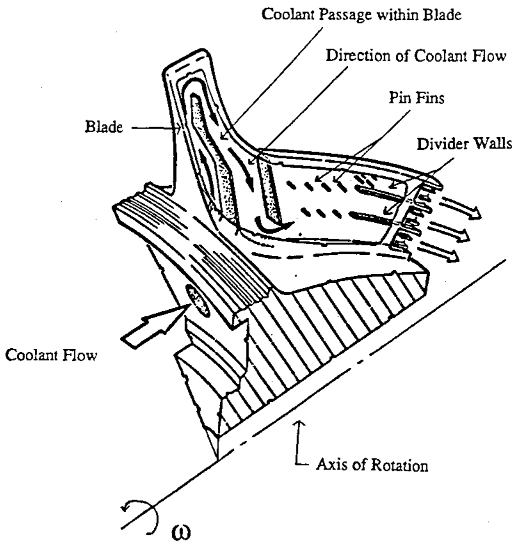

Fig. 1 Schematic of a turbine blade with a coolant passage inside it.

ployed for the coolant-passage problem are examined in relation to their cost-effectiveness. Afterwards, a grid structure deemed most suitable for the coolant-passage problem is proposed. Finally, that proposed grid structure is demonstrated by applying it to compute the flow within two different coolant passages.

Grid Structure and Cost-Effectiveness

All grids can be classified as structured, unstructured, or mixed (i.e., a combination of structured and unstructured). Structured grids can further be classified as single- or multiblock grids. Depending upon how the different single-block grids are combined together to form the multiblock grid, the multiblock grid can be "completely continuous," "partially continuous," "partially discontinuous," or "completely discontinuous."[11,12] As used here, structured grids are those that employ quadrilateral or hexahedral cells with regular neighbors, and unstructured grids are those that employ triangular or tetrahedral cells with irregular neighbors. The reason for requiring unstructured grids to use triangles or tetrahedra in addition to having irregular neighbors is that a grid system with quadrilaterals or hexahedra with occasional irregular neighbors can be taken to be a multiblock structured grid.

The most important advantage of unstructured grids over structured ones is that they are the easiest to generate, and local grid refinement can readily be made. However, there are three major disadvantages associated with their use. The first disadvantage is that unstructured grids, in general, have and require more cells than structured grids. In fact, even when the same set of grid or nodal points are used, unstructured grids have twice as many cells as structured grids, since there are two triangles in a rectangle and two tetrahedra in a hexahedron. For high Reynolds number shear flows over streamlined surfaces, the required number of cells can be considerably more. This is because the angles between intersecting lines or planes of triangular or tetrahedral cells cannot be too small, even though the length scales in the streamwise direction can be orders of magnitude larger than those in directions normal to it, based on fluid mechanics considerations. The second disadvantage is that, since the connectivity between grid or nodal points is random, efficient solution algorithms, such as those based on line iterative processes [e.g., alternating-direction implicit (ADI), and lower-upper (LU) techniques] developed for structured grids, cannot be used effectively. The third disadvantage is that, since the cells are triangular or tetrahedral, grid lines cannot be aligned with the flow direction. This implies that true upwind schemes would be needed to capture physics. It also implies that higher numerical diffusion would result unless cell aspect ratios are kept near unity with cell sizes on the order of the shortest length scales that need to be resolved. For structured grids, grid alignment is possible in most situations, except those involving recirculating flows where the grid aspect ratio also needs to be kept near unity to minimize numerical diffusion.

Dawes[13] successfully used the unstructured-grid approach to study the flow within a realistic coolant passage. The coolant passage that he studied was in a radial turbine blade with all of the attendant geometric complexities. To overcome the difficulty associated with unstructured grids' need for a large number of grid points next to shear layers, walls functions were employed so that grid spacings next to solid walls can be larger. The numerical method employed to obtain solutions was an explicit method (second-order-accurate Runge-Kutta with residual smoothing). An explicit method was used because the random connectivity associated with unstructured grids makes implicit methods difficult to implement. Despite the less efficient numerical method of solution and a larger number of grid points, the unstructured-grid approach of Dawes is highly competitive in terms of the total amount of time and cost needed to generate a solution. This is because the amount of time required to generate a grid for the coolant passage is considerably less than the one man-month mentioned earlier, in fact, only a fraction of it.

For structured grids, the major advantages over unstructured grids are as follows. First, very efficient solution algorithms exist because of the regularity in the connectivity between grid points (e.g., ADI and LU). And, second, far fewer grid points are needed for high Reynolds number shear flows since, for structured grids, grid spacings in the streamwise direction can be orders of magnitude larger than those in directions normal to it as long as grid lines are aligned with the flow direction, a condition that is met easily by boundary-fitted grids along solid surfaces. The main disadvantage associated with structured grids is that they can be much more difficult to generate. This difficulty is greatest for multiblock grids that are completely continuous and partially continuous, less so for those that are partially discontinuous, and least for those that are completely discontinuous.

Simulations of the coolant-passage problem using structured grids were performed by Steinthorsson et al.[8] and Ippoloto.[14] Both of these studies investigated the three-dimensional flowfield in the same coolant passage as Dawes.[13] Steinthorsson et al. employed a partially continuous multiblock grid, whereas Ippoloto employed a partially discontinuous one. For both of these grid structures, the grid generation process alone can take up to one man-month or more to complete, which is considerably longer than that required by the unstructured-grid approach. Thus, the structured-grid approaches attempted so far are clearly not competitive when compared with the unstructured-grid approach in terms of timeliness and cost-effectiveness.

Since structured grids do have a number of advantages over unstructured ones, this investigation seeks to identify a structured-grid approach that can be competitive with the unstructured-grid approach for the coolant-passage problem. In this regard, completely discontinuous structured grids (i.e., structured grids that can overlap each other) known as the chimera grids are worth investigating. This is because they are the easiest of the structured grids to generate. In fact, the ease of generating such grids is comparable to that of unstructured grids. And, under certain situations, that ease can even be superior than that for unstructured grids. For example, if a bank of pin fins in a coolant passage needs to be moved from one location to another, unstructured grids will require the generation of a totally new grid or at least a major portion of it. But, in the case of chimera grids, it is only necessary to move the grids about the bank of pin fins that were generated earlier from the old location to the new location without the need to generate any new grids. Thus, chimera grids rival unstructured grids in terms of cost-effectiveness in grid generation under such situations. Moreover, chimera grids surpass unstructured grids when it comes to efficient solution algorithms since they are made up of structured grids.

Chimera Grids

The many attractive features of chimera grids, cited in the previous section, make them highly desirable for use in the study of multidimensional problems with complex geometries. Here, it is contended that an approach utilizing chimera grids may be the most suitable one for use in the analysis and design of coolant passages, especially those in radial-turbine blades such as the one shown in Figs. 1 and 2. The reasons are as follows:

1) For coolant passages in radial-turbine blades, the major objective of the design process is to optimize the placement of baffles and pin fins to give rise to the most uniform flow and heat transfer possible over all parts of the coolant passage with minimal recirculation regions and no hot spots. With a chimera-grid approach, this task of optimizing the location of the baffles and pin fins can be achieved easily by simply moving the grids about the baffles and the pin fins from one location to another.

2) Since grids can overlap in chimera grids, the domain regardless of its geometric complexity can be partitioned and molded into sub-domains with simple geometries such as rectangles and circles or whatever shape that is most convenient for the grid generation process. For each of these geometrically simplified subdomains, a grid can easily be generated. Thus, the generation of a complete grid for even an extremely complex-shaped domain can be made simple and straightforward.

3) Chimera grids can easily be used to eliminate mapping singularities associated with O-O, O-H, C-H, . . . structured grids by patching H-H grids over the singularities.[15]

Application to Turbine-Blade Coolant Passages

To demonstrate the feasibility and usefulness of the chimera-grid approach for the analysis and design of flows in coolant passages, two different coolant passages were studied. The coolant passage problems studied along with their formulation, as well as generation of chimera grids, computations on chimera grids, and results obtained are described below.

Description of Test Problems

The two coolant passages analyzed in this study are as follows: The first coolant passage analyzed is one inside a radial turbine component with 13 blades. A schematic diagram of one blade in this turbine component is shown in Fig. 1. Also shown in that figure is a cutaway view of the coolant passage within the blade and the direction of coolant flow. Figure 2 shows the projection of that coolant passage in a r-z plane.

For the coolant passage shown in Figs. 1 and 2, the coolant was air, and its stagnation temperature T_c and pressure P_c in a frame of reference fixed on the blade were 243.3 K and 168,900 Pa, respectively. The static pressures P_b at the exit of the coolant passage was 61,400 Pa. Even though in practice the coolant-passage wall temperature varies with position, in this study it was taken to be a constant at 300 K.

The second coolant passage studied is shown in Fig. 3. This idealized coolant passage was studied because experimental data are being gathered for it.[16] The experimental data gathered can be used to validate the computational tool developed here. For this coolant passage, the coolant was air with a stagnation temperature of 294 K and a stagnation pressure of 100,000 Pa. The back pressure at the exit was adjusted to match the flow rates used in the experiments.

Formulation of Test Problems

The flow within the two coolant passages described in the previous section was modeled by the conservation equations of mass, momentum ("full compressible" Navier-Stokes), and total energy, written in generalized coordinates and cast in strong conservation-law form. Since the goal of this study is to demonstrate the usefulness of chimera grids for computing flows inside coolant passages, effects due to rotation were not considered at this time. Also, only some effects due to turbulence were accounted for. The Baldwin-Lomax algebraic turbulence model[17] was used to increase the effective diffusivities in boundary layers next to all solid walls, except those next to pin fins and baffles.

Numerical Method of Solution

Solutions to the conservation equations of mass, momentum, and total energy described in the previous section were obtained by using a modified version of the OVERFLOW code.[18] The modified version of OVERFLOW is given the name OVERFLOW.coolant. OVERFLOW.coolant uses a partially split, two-factored, finite-volume algorithm.[18,19] All inviscid flux terms along the ξ direction were upwind differenced by using the flux-vector splitting procedure of Steger and Warming.[20] Inviscid flux terms in directions normal to the ξ direction were centrally differenced, as were diffusion terms in all directions. The set of grid lines in the ξ direction

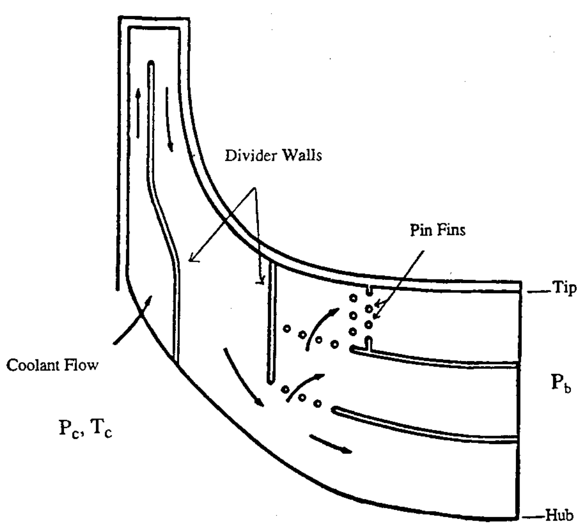

Fig. 2 Schematic of the coolant passage in an r-z plane.

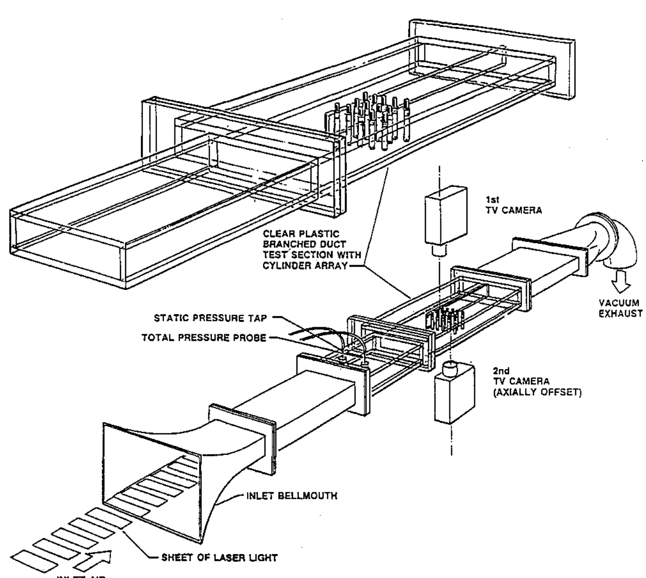

Fig. 3 Schematic diagram of a coolant passage where experimental data is being gathered.

is boundary conforming and is distributed to align with the general flow direction as much as possible.

Generation of Chimera Grids

In order to obtain solutions by using OVERFLOW.coolant or any other code, the domains of the problems shown in Figs. 1–3 must be replaced by grid systems. As noted earlier, the generation of partially continuous and partially discontinuous structure multiblock grids for these domains is extremely tedious and time-consuming because of the baffles and the large number of pin fins. However, if the different grids in a structured multiblock grid are allowed to overlap in an arbitrary manner as in chimera grids, then the grid-generation process can become quite simple. Why this is so will become clear after going through the steps of generating a chimera grid.

Regardless of the complexity of the geometry, chimera grids can be generated by going through the following four steps: 1) identify the key components of the geometry, 2) decide on the ideal grid structure for each component, 3) decide on which grid or grids are to cover or fill the entire domain, and 4) decide on the amount of overlap in different grids. These four steps are illustrated below by generating a chimera grid for the coolant passage shown in Figs. 1 and 2.

Step 1

For the domain shown in Figs. 1 and 2, the key components are as follows: pin fins (12 in number, all identical in diameter but differ slightly in length), baffles (three in number and differ in geometry), coolant passage without the pin fins and baffles.

Step 2

The ideal grid structure for each component identified in the first step is as follows. For the pin fins, the ideal grid structure is an O-H grid. Only one such grid needs to be generated since all 12 pin fins are identical in diameter. For the baffles, the ideal grid is a C-H grid that wraps around each baffle. Here, three different grids need to be generated. For the coolant passage without the pin fins and baffles, the ideal grid is a H-H grid. For this component, it is noted that the flow-cross-sectional area changes considerably after the U-bend. Thus, to minimize the total number of grid points needed, two overlapping H-H grids could be used, one H-H grid with fewer grid points in the cross section for the narrower part of the passage and another H-H grid with more grid points in the cross section for the wider part of the passage.

The grid generated for each of these components is shown in Fig. 4. Note that the generation of these grids is rather straightforward since the geometry of each component has been made simple. Also, note that all grids generated are three-dimensional (although not shown as such), and are clustered next to solid walls in order to resolve the sharp flow gradients there.

Step 3

For the coolant-passage problem studied here, it is easy to decide which grid or grids should fill the entire domain of interest. Clearly, it should be the two H-H grids that span the entire coolant passage. With such a choice, the baffles and pin fins can be placed anywhere within it. Also, even after the pin fins and baffles have been placed, they can be removed and then located elsewhere without the need to generate new grids.

Step 4

In this step, all grids generated for each component are overlapped on top of each other and/or the grid or grids that fill the entire domain. For the coolant-passage problem, the result of this overlapping process is shown in Fig. 5. Note that in regions where grids overlap, grid spacings were made comparable in order to minimize aliasing errors. Not shown in Fig. 5 is the boundary of each grid. Information from one grid is transferred to the grid or grids that it overlaps through grid boundaries by trilinear interpolation. Since trilinear interpolation can produce conservation errors and cause gradients to shift, an alternative way of transferring information between grids is to use unstructured-grid formulation in the overlapped regions.

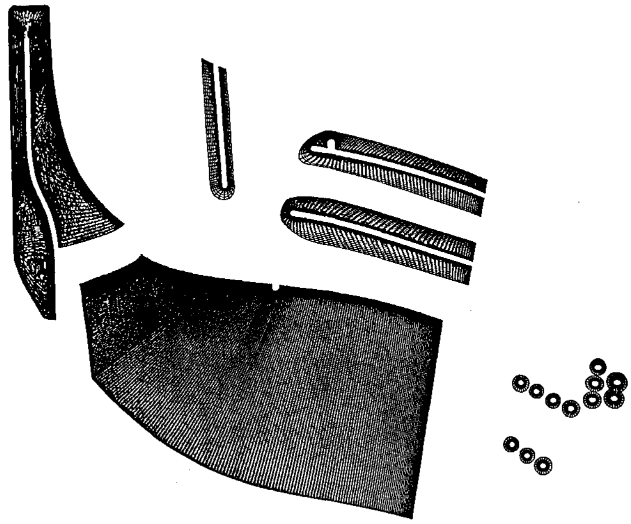

Fig. 4 Grid generated about each component of the coolant passage shown in Figs. 1 and 2.

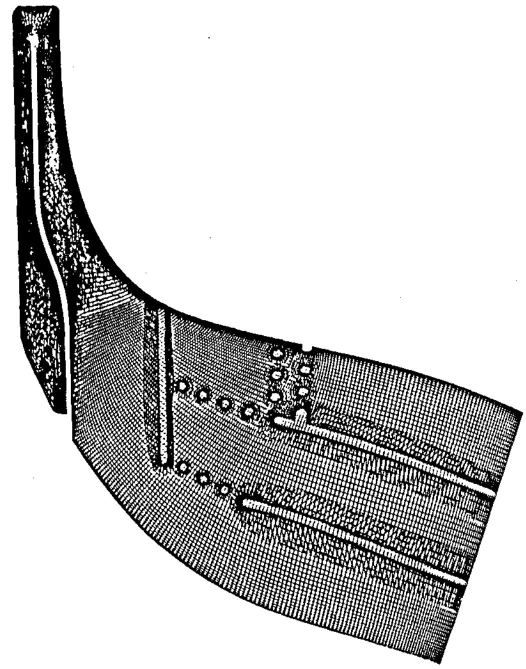

Fig. 5 Chimera grid generated for the coolant passage shown in Figs. 1 and 2.

This completes the generation of a chimera grid for the coolant passage shown in Figs. 1 and 2. As a summary, the chimera grid shown in Figs. 4 and 5 consists of 17 overlapping grids and has a total of 696,473 grid points. A chimera grid for the coolant passage shown in Fig. 3 can be generated in a manner similar to the one just described. A chimera grid thus generated is shown in Fig. 6. It also consists of 17 overlapping grids and has a total of 889,400 grid points. Since the focus of this study is not on the detailed flow physics inside coolant passages, further details about the grids generated (e.g., minimum and maximum grid spacings that are needed to assess the results) are not given.

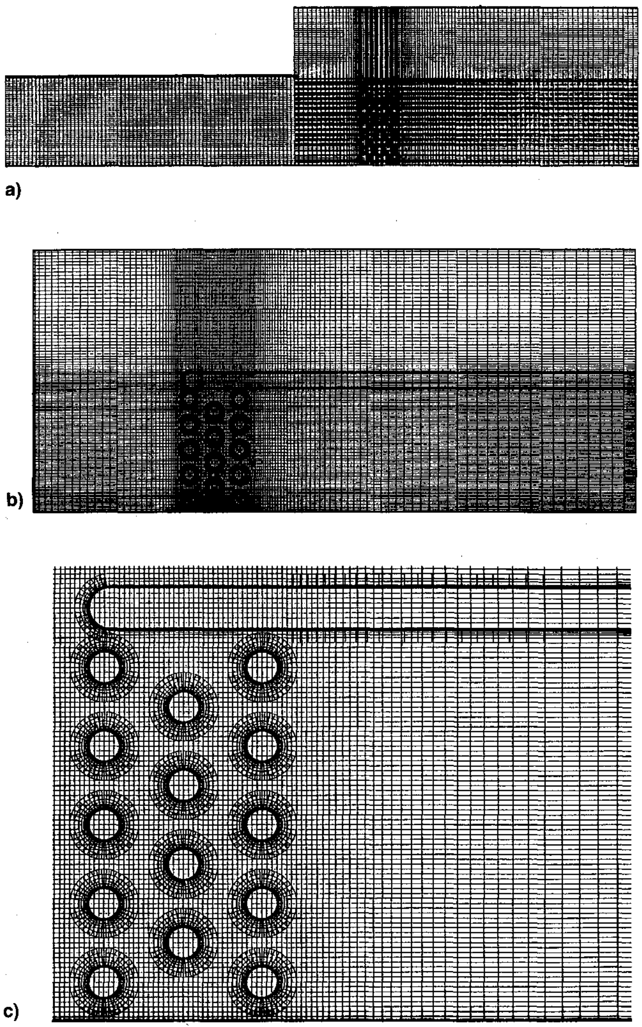

Fig. 6 Chimera grid generated for the coolant passage shown in Fig. 3: a) entire chimera grid, b) one close-up view, and c) another close-up view.

At this point, it is noted that the coolant passage shown in Figs. 1 and 2 is geometrically complex in three dimensions, but the one shown in Fig. 3 is geometrically complex only in two dimensions. For the former, each grid in the chimera grid must be generated as a three-dimensional grid. For the latter, each grid could be generated as a three-dimensional grid directly, or as a two-dimensional grid first and then stacked together to form a three-dimensional grid.

By following the procedure described above for generating chimera grids, one can see that chimera grids greatly simplify the grid generation process, which in turn greatly reduces the amount of man hours needed to setup the grid. For the two coolant passages studied here, each chimera grid can be generated within a few days. This should be contrasted with the one or more man months required to generate partially continuous and partially discontinuous structured grids for the same geometries. Also, once a chimera grid has been generated, only about 1–2 man hours are needed to move one or more pin fins or baffles from one location to another. This is another order of magnitude reduction in man hours needed to setup a grid.

Computations on a Chimera Grid

During computations of the flow in both coolant passages, the flowfield in each grid was analyzed one at a time. For the coolant passage in a radial-turbine blade, the following order was used: the H-H grid in the entrance region, the H-H grid in the rest of coolant passage, the C-H grids about the three baffles, and finally the O-H grids about the 12 pin fins. For the other coolant passage (Fig. 3), the order was as follows: the H-H grid in the entrance region, the H-H grid in the rest of the domain, and the O-H grids about the 15 pin fins. This process of analyzing the flow in one grid at a time until all grids are analyzed is repeated for each time step until a converged solution is obtained. Here, a solution is assumed to be converged if the second norm of the residual levels out for at least 500 time steps. Typically, at that time, the second norm is about 10^{-9}. Here, it is noted that the residual oscillates about some average value as it levels out, and the amplitude of those oscillations was less than 10^{-9} when the residual was 10^{-9}.

Information from one grid was passed to another grid in the overlapped region via trilinear interpolation at grid boundaries. The required interpolation coefficients were obtained by using the PEGSUS code.[21] The details of the interpolation process used to transfer data from one grid to another can be found in Ref. 21, and will not be repeated here. However, it is important to note that although the interpolation procedure used was found to be stable and to lead to convergent solutions, some conservation errors were introduced. For the coolant passage problems investigated here, the conservation errors were found to be small (less than 2%).

Results

Results were obtained for the flow and heat transfer inside both coolant passages described earlier, and depicted in Figs. 1–3 to demonstrate that chimera grids can indeed be used to obtain solutions. Some computed results for the flow in the coolant passage inside the radial-turbine blade are shown in Figs. 7 and 8. Figure 7 shows the velocity vector field and contours of Mach number, pressure, and temperature in a twisted plane midway between the suction- and pressure-side walls of the coolant passage. Also shown in Fig. 7 is the contour of the heat transfer coefficient on the suction- and pressure-side walls. Figure 8 shows the velocity vector field and contours of pressure and temperature in a cross section that is located in the middle of the coolant passage's U-bend as seen from the downstream side. Thus, the right and left walls correspond respectively to the outer and inner radius of the U-bend.

These results show that the computations were able to predict the separated regions where expected. For example, a separated region can be observed at the entrance of the coolant passage, around the U-bend, and in other regions of the flowfield where adverse pressure gradients were sufficiently large (Fig. 7a). Also, secondary flows due to curvature of the coolant passage in the U-bend were predicted (Fig. 8). The separated regions around and downstream of the U-bend were found to constrict the flow considerably and caused the Mach number in that part of the coolant passage to approach 0.9. The expected vortex shedding behind each pin fin was not captured by the computations. This is because an insufficient number of grid points were placed around each pin fin to resolve those physics.

Figure 9 shows the contours of Mach number, pressure, and temperature computed in the midplane of the coolant passage shown in Fig. 3. Figure 9 also shows the heat transfer coefficient on a wall of the coolant passage.

The heat transfer coefficient shown in Fig. 9d can be compared with the measured values[16] given in Fig. 10. Such a comparison shows that the computed and measured heat transfer coefficient are in reasonable agreement. Basically, they have the same order of magnitude and are qualitatively similar. The differences between the computed and measured values can be attributed to several sources. First, the flow rates in the computed and measured coolant passages are different by 12%. Second, efforts were not made to refine the grid or the turbulence model since the focus of this study was to demonstrate the utility of the chimera grids for the

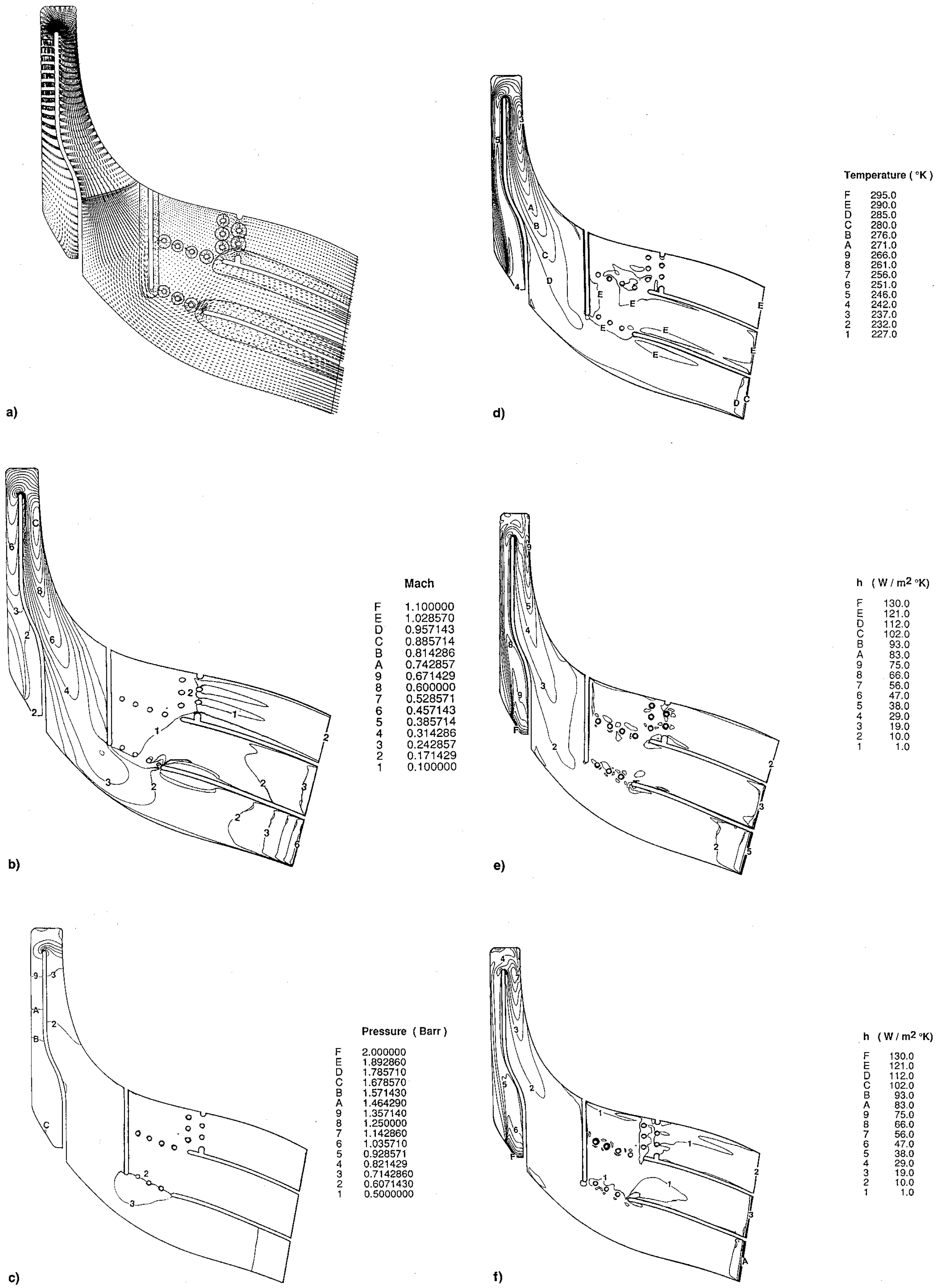

Fig. 7 Computed flow and heat transfer in coolant passage shown in Figs. 1 and 2: a) velocity, b) Mach number contours, c) pressure contours, d) temperature contours, e) heat transfer coefficient on suction-side wall, and f) heat transfer coefficient on pressure-side wall.

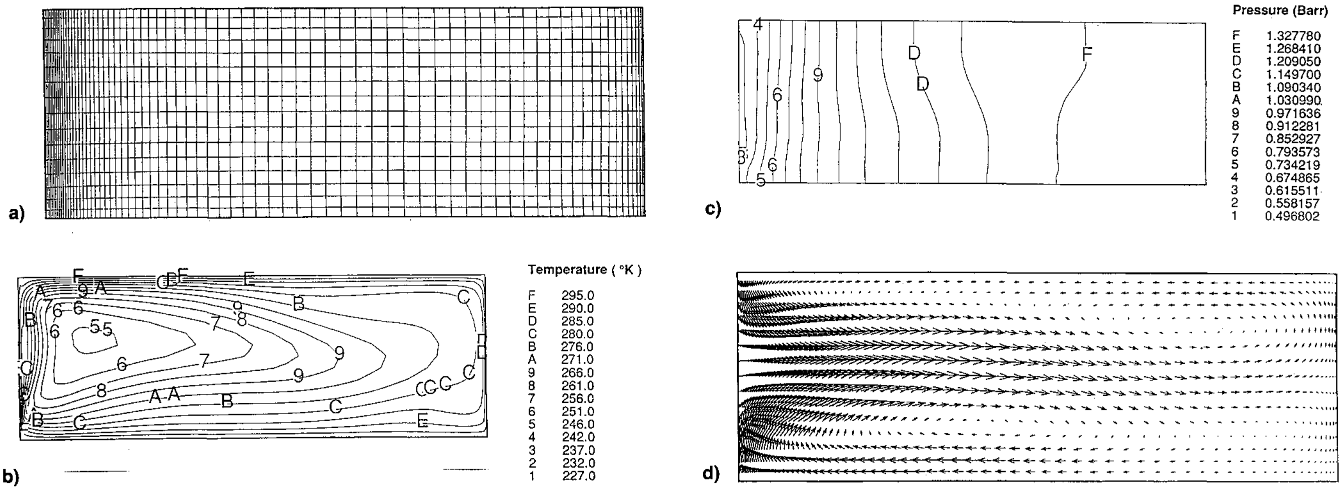

Fig. 8 Computed results in a cross section at middle of U-bend in the coolant passage shown in Figs. 1 and 2: a) grid, b) velocity, c) pressure contours, and d) temperature contours.

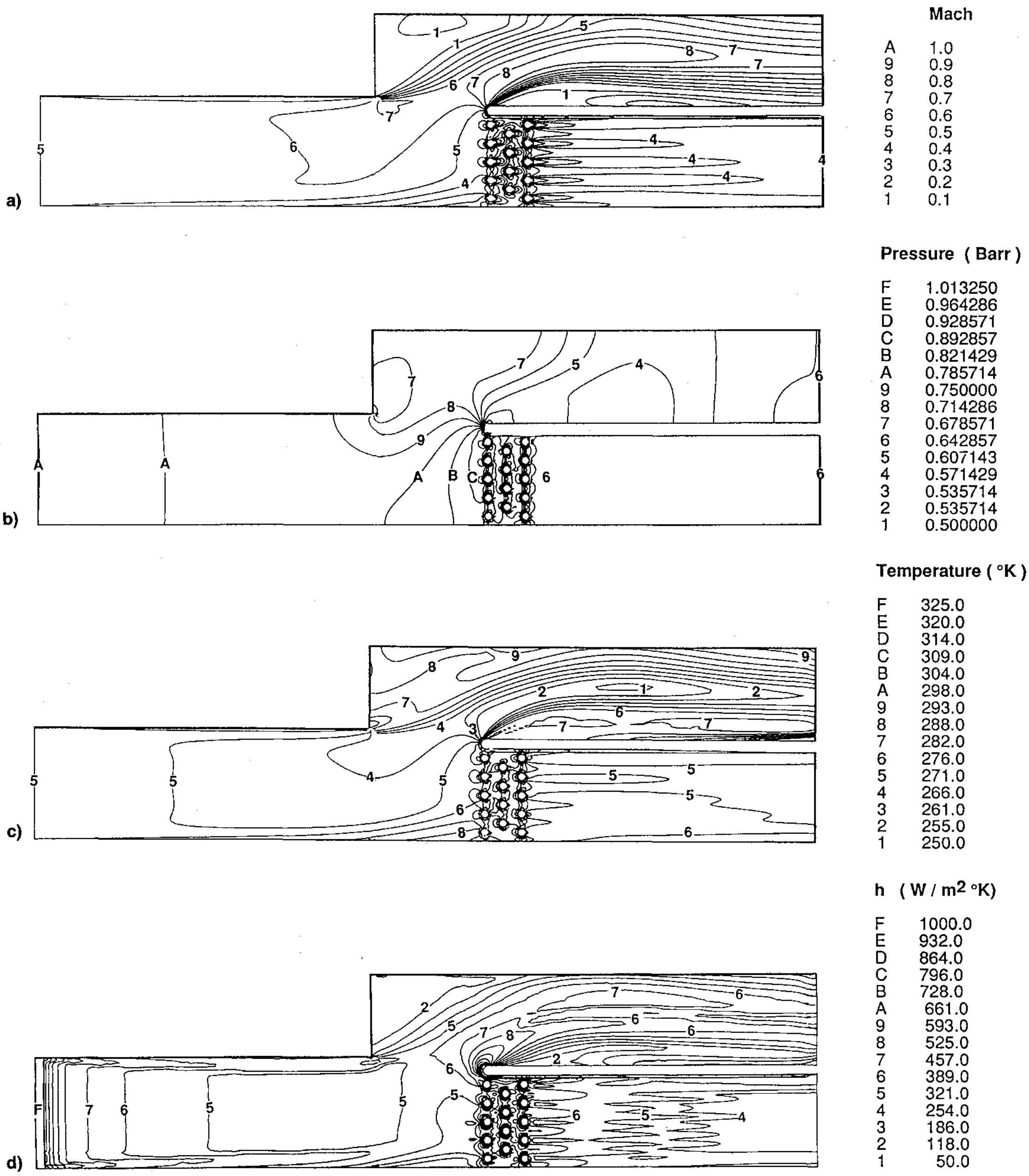

Fig. 9 Computed flow and heat transfer in the coolant passage shown in Fig. 3: a) Mach number contours, b) pressure contours, c) temperature contours, and d) heat transfer coefficient.

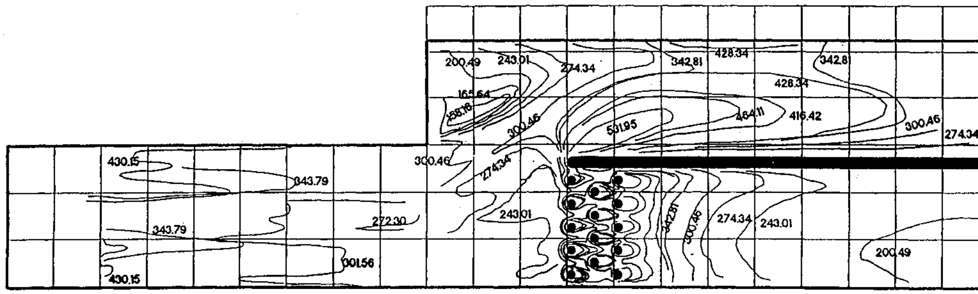

Fig. 10 Experimentally measured heat transfer coefficient for the coolant passage shown in Fig. 3 (obtained from Ref. 16).

coolant passage problem and not to study the physics of the flow. It is for this reason that only minimal discussions were given for the results presented in Figs. 7–9.

Although discussion of the results shown in Figs. 7–9 were minimal, their inspection does show that the results obtained are reasonable. This indicates that chimera grids are indeed useful for obtaining solutions in geometries as complex as those in coolant passages.

Summary

In this article, the overlapping structured grid known as the chimera grid was shown to be ideally suited for application to the design and analysis of coolant passages with highly complex geometries. The reasons are as follows: first, chimera grids greatly reduce the number of man hours needed to generate a grid. Second, parametric studies to optimize baffle and pin fin locations can be performed without generating any new grids. Thus, those parametric studies can be performed in a highly cost-effective manner with rapid turnaround time. And third, since chimera grids utilize structured grids, very efficient relaxation schemes such as ADI and LU methods can be used to accelerate convergence rates. This reduces the amount of computer time needed to generate solutions, and thereby further reducing the overall time and cost needed to obtain solutions.

Acknowledgments

This work was supported by Grant NAG 3-929 from NASA Lewis Research Center. The computer time was provided by the NAS facility. The authors are grateful for this support.

References

[1]Suo, M., "Turbine Cooling," *Aerothermodynamics of Aircraft Engine Components*, edited by C. Oates, AIAA, New York, 1985, pp. 275–328.

[2]Taylor, J. R., "Heat Transfer Phenomena in Gas Turbines," American Society of Mechanical Engineers Paper 80-GT-172, 1980.

[3]Chyu, M. K., and Wu, L. X., "Combined Effects of Rib Angle-of-Attack and Pitch-to-Height Ratio on Mass Transfer from Surface with Transverse Ribs," *Experimental Heat Transfer*, Vol. 2, 1989, pp. 291–308.

[4]Kumar, G. N., Roelke, R. J., and Meitner, P. L., "A Generalized One Dimensional Computer Code for Turbomachinery Cooling Passage Flow Calculations," NASA TM 102079, July 1989.

[5]Snyder, P. H., and Roelke, R. J., "Design of an Air-Cooled Metallic High-Temperature Radial Turbine," *Journal of Propulsion and Power*, Vol. 6, No. 3, 1990, pp. 283–288.

[6]Chyu, M. K., "Regional Heat Transfer and Pressure Drop in Two-Pass and Three-Pass Flow Passages with 180-Degree Sharp Turns," *Journal of Heat Transfer*, Vol. 113, No. 1, 1991, pp. 63–70.

[7]Prakash, C., and Zerkle, R., "Prediction of Turbulent Flow and Heat Transfer in a Radially Rotating Square Duct," *Journal of Turbomachinery*, Vol. 114, No. 4, 1992, pp. 835–846.

[8]Steinthorsson, E., Shih, T. I-P., and Roelke, R. J., "Computations of the Three-Dimensional Flow and Heat Transfer Within a Coolant Passage of a Radial Turbine Blade," AIAA Paper 91-2238, June 1991.

[9]Steinthorsson, E., Shih, T. I-P., and Roelke, R. J., "Algebraic Grid Generation for Coolant Passages of Turbine Blades with Serpentine Channels and Pin Fins," AIAA Paper 91-2366, June 1991.

[10]Steinthorsson, E., personal communication, ICOMP, NASA Lewis Research Center, Cleveland, OH, 1991.

[11]Eiseman, P. R., and Erlebacher, G., "Grid Generation for the Solution of Partial Differential Equations," NASA CR-178365, Aug. 1987.

[12]Shih, T. I-P., Bailey, R. T., Nguyen, H. L., and Roelke, R. J., "Algebraic Grid Generation for Complex Geometries," *International Journal for Numerical Methods in Fluids*, Vol. 13, 1991, pp. 1–31.

[13]Dawes, W. N., "The Solution-Adaptive Numerical Simulation of the 3D Viscous Flow in the Serpentine Coolant Passage of a Radial Inflow Turbine Blade," American Society of Mechanical Engineers Paper 92-GT-193, June 1992.

[14]Ippolito, J., "Three-Dimensional Flow Analysis of the Internal Passage of a Cooled Radial Rotor," Rept. 30-02-004, ADAPCO, Melville, NY, April 1991.

[15]Shih, T. I-P., Rimlinger, M. A., and Chyu, W. J., "Three-Dimensional Shock-Wave/Boundary-Layer Interactions with Bleed," *AIAA Journal*, Vol. 31, No. 10, 1993, pp. 1819–1826.

[16]Russell, L. M., Thurman, D. R., and Simonyi, P. S., "Measurements and Computational Analysis of Heat Transfer and Flow in a Simulated Turbine Blade Internal Cooling Passage," AIAA Paper 93-1797, June 1993.

[17]Baldwin, B., and Lomax, H., "Thin Layer Approximation and Algebraic Model for Separated Turbulent Flows," AIAA Paper 78-257, Jan. 1978.

[18]Buning, P. G., and Chan, W. M., "OVERFLOW/F3D User's Manual," NASA Ames Research Center, 1991.

[19]Steger, J. L., Ying, S. X., and Schiff, L. B., "A Partially Flux-Split Algorithm for Numerical Simulation of Compressible Inviscid and Viscous Flow," *Proceedings of the Workshop on Computational Fluid Dynamics*, Inst. of Nonlinear Sciences, Univ. of California, Davis, Davis, CA, 1986.

[20]Steger, J. L., and Warming, R. F., "Flux-Vector Splitting of the Inviscid Gasdynamic Equations and Application to Finite-Difference Methods," *Journal of Computational Physics*, Vol. 40, No. 2, 1981, pp. 263–293.

[21]Benek, J. A., Buning, P. G., and Steger, J. L., "A 3-D Chimera Grid Embedding Technique," AIAA Paper 85-1523, July 1985.