IN-34
6251
P. 12

NASA Contractor Report 198419
ICOMP–95–23; AIAA–95–0177

Computations of Viscous Flows in Complex Geometries Using Multiblock Grid Systems

Erlendur Steinthorsson
Institute for Computational Mechanics in Propulsion
Lewis Research Center
Cleveland, Ohio

and

Ali A. Ameri
AYT Corporation
Beachwood, Ohio

(NASA-CR-198419) COMPUTATIONS OF VISCOUS FLOWS IN COMPLEX GEOMETRIES USING MULTIBLOCK GRID SYSTEMS (NASA. Lewis Research Center) 12 p

N96-15124

Unclas

G3/34 0079431

Prepared for the
33rd Aerospace Sciences Meeting and Exhibit
sponsored by the American Institute of Aeronautics and Astronautics
Reno, Nevada, January 9–12, 1995

National Aeronautics and
Space Administration

INSTITUTE FOR COMPUTATIONAL MECHANICS IN PROPULSION
NASA
ICOMP
Lewis Research Center
Ohio Aerospace Institute

COMPUTATIONS OF VISCOUS FLOWS IN COMPLEX GEOMETRIES USING MULTIBLOCK GRID SYSTEMS

Erlendur Steinthorsson*
Institute for Computational Mechanics in Propulsion,
NASA Lewis Research Center, Cleveland, Ohio

and

Ali A. Ameri†
AYT Corporation, Beachwood, Ohio

Abstract

Generating high quality, structured, continuous, body-fitted grid systems (multiblock grid systems) for complicated geometries has long been a most labor-intensive and frustrating part of simulating flows in complicated geometries. Recently, new methodologies and software have emerged that greatly reduce the human effort required to generate high quality multiblock grid systems for complicated geometries. These methods and software require minimal input from the user—typically, only information about the topology of the block structure and number of grid points. This paper demonstrates the use of the new breed of multiblock grid systems in simulations of internal flows in complicated geometries. The geometry used in this study is a duct with a sudden expansion, a partition and an array of cylindrical pins. This geometry has many of the features typical of internal coolant passages in turbine blades. The grid system used in this study was generated using a commercially available grid generator. The simulations were done using a recently developed flow solver, TRAF3D.MB, that was specially designed to use multiblock grid systems.

Introduction

Generating high quality, structured, continuous, body-fitted grid systems (multiblock grid systems) for complicated geometries has long been a most labor-intensive and frustrating part of simulating flows in complicated geometries. Although the grid generation techniques that have been used are often termed "automatic,"[1] they have in fact been semi-automatic at best since considerable human involvement has been required to generate acceptable grids. The human involvement has been to "carve up" the flow domain into zones of reasonable shape and, through trial and error, set various parameters for the grid generators that optimize the grid system with respect to the conflicting objectives of (a) clustering grid points where they are needed, (b) maximizing "smoothness" (e.g., minimize rapid changes in grid spacing), and (c) minimizing skewness. To avoid the difficulty in generating body-fitted, structured grid systems, researchers have invented different approaches to the "gridding" of flow domains. Using separate body-fitted grids for different parts of the geometries and letting the component grids overlap in an arbitrary manner (Chimera grids[2–4]), has been a popular choice (e.g., Ref's 5–7). The use of unstructured grids has also received much attention in recent years, precisely for the relative automation of the grid generation process for complex geometries and for the apparent flexibility that unstructured grids offer to dynamically adapt the grid to the computed solutions.

Recently, new methodologies and new software[8–12] have emerged that greatly reduce the human effort required to generate high quality multiblock grid systems for complicated geometries. These methods and software require minimal input from the user—typically, only information about the topology of the block structure and density of grid points. The grid generation software then optimizes the grid with respect to orthogonality, smoothness and other conflicting objectives. A distinguishing feature of the multiblock grid systems is that they, in general, cannot be mapped to a single index (or computational) space. In essence, these multiblock grid systems are unstructured at the block level but structured at the cell level (array arrangement of cells within each block). The absence of stucture at the block level results in great flexibility in the creation of mesh topologies for complicated geometries and an opportunity to generate high quality grid systems. The only requirement for the block structure is that the blocks be topologically quadrilateral in two dimensions (2D) and hexahedra in three dimensions (3D).

*Senior Research Associate, Member AIAA
†Research Engineer, Member AIAA.

The objective of this paper is to demonstrate the use of the new breed of multiblock grid systems in simulations of internal flows in complicated geometries. The geometry used in this study is a duct with a sudden expansion, a partition and an array of cylindrical pins (see Fig. 1). This geometry has many of the features of internal coolant passages in turbine blades and has been used in experiments aimed at increasing the understanding of coolant flow and heat transfer in turbine blades[13]. The grid system used in this study was generated using a commercially available grid generator called GridPro.[12] The simulations were done using a recently developed flow solver, TRAF3D.MB, that was specially designed to use multiblock grid systems[14].

The paper is organized as follows: Following this introduction, the problem to be studied is described. Afterwards, the grid system used in the computations is discussed and some important features pointed out. Next, the major elements of flow solver will be briefly described. Finally, results of computations are presented. The paper ends with some concluding remarks.

Description of Problem

The problem to be analyzed is the flow of air through a duct with a sudden expansion and, downstream of the expansion, a partition that splits the duct in two branches and an array of cylindrical pins. A schematic drawing of the duct is shown in Fig. 1. As indicated in the figure, the air flows through the duct from left to right. The array of pins below the partition creates a blockage to the flow in the lower branch of the duct that acts to divert some of the flow through the upper branch. The floor of the duct is heated.

The flow field in the branched duct is quite complicated. Nonetheless, some qualitative features of the flow are predictable based on experience and intuition. In particular, separation of the flow at the expansion corner and on the upper side of the partition is expected. However, the location of reattachment points on the upper side wall and partition, the extent of the recirculation region above the partition, and the effect of the pin array on the flow field cannot be predicted easily without numerical simulations. Consequently, heat transfer and wall temperature patterns in the duct cannot be reliably predicted without detailed numerical investigations.

The branched duct geometry used in the current study has many of the features found in coolant passages of turbine blades, such as the pins and the sudden expansion. Also, the heating of the floor emulates the thermal loading on a turbine blade. The duct was designed and used in experiments at NASA Lewis Research Center[13] in a project aimed at enhancing understanding of flows in turbine blade coolant passages. In the experiments, measurements of flow field and heat transfer were done for inflow Reynolds numbers of 45,000, 335,000, and 726,000 (based on flow speed in the entrance section and hydraulic diameter of the inflow duct), corresponding to Mach number from 0.03 to 0.45. Heat transfer results from the experiments were reported in Ref. 13.

Grid System

A multiblock grid system was generated for the branched duct geometry shown in Fig. 1. The grid system was generated using a commercially available grid generator called GridPro.[12] A 2-D grid system was initially generated for the cross section shown in Fig. 1b. A stack of the 2-D grids forms the final 3-D grid system, which extends from the channel floor to the symmetry plane midway between the floor and the ceiling. Figures 2–4 show the 2-D grid system used in the current study. Figure 2a shows the block structure of the grid whereas Fig. 2b shows a close-up view of the block structure around the partition and pins. Figure 3 shows grid system for the main-part of the duct and Fig. 4 shows a close-up view of the grid near the corner at the sudden expansion and around the partition and pins.

There are several features to note about the grid system used in this study. First, as seen in Fig. 2a, there are several singular points in the grid where three, five or six blocks come together. Because of these singular points, the grid system cannot be mapped to a single computational space, and flow solvers utilizing grid systems of this type must be set up to handle the singularities properly. However, allowing these kinds of singularities in the grid is exactly what gives the flexibility that is needed for generating high quality continuous structured grids for complex geometries such as the branched duct. A second feature to note is that high quality "boundary-layer type" grids are obtained around all solid surfaces. Thus, O-grids are formed around the pins and grid lines wrap around the partition (see Fig. 4a). Also, grid lines run parallel to the side walls of the duct and turn around the slightly rounded expansion corner (note, in the actual geometry used in the experiments reported in Ref. 13, the expansion corner is slightly rounded, here modeled with a radius of 0.5mm). These high quality boundary-layer type grids lend themselves well to computations of viscous flows and facilitates accurate implementation of physical boundary conditions. A final feature to note about the grid system in Fig. 2–4 is that the grid everywhere smooth and, except in the immediate vicinity of singularities, nearly orthogonal. In particular, the grid is very nearly orthogonal at all solid surfaces, as the singularities are located away from all physical boundaries.

The 3-D grid system used in the current study contains about 2 million cells. Each 2-D cross section of the grid contains about 50,000 grid points, 65% of which are in the blocks immediately adjacent to solid wall surfaces (the boundary layer grids). The grid system was generated in 283 full-face matching blocks. The size of the 2-D blocks varies from 8×8 to 65×24. Typical grid spacing normal to sidewalls, pins and partition is $2 \cdot 10^{-4}$ whereas grid spacing at the channel floor is $1.1 \cdot 10^{-4}$ (note, dimensions are normalized by the height from the floor of the channel to the symmetry plane).

Flow Solver

The simulations performed in this study were done using a recently developed computer code called TRAF-3D.MB.[14] This code is is a general purpose flow solver, designed specifically for use in simulations of flows in complicated geometries. The code is based on an earlier code, TRAF3D, an efficient code designed for simulations of flows in turbine cascades.[15] The TRAF3D.MB code uses a cell-ceneterd finite volume discretization of the compressible Navier-Stokes equations. The discretization is second-order accurate in space. The code offeres a choice between central differencing with artificial dissipation and the AUSM[16] scheme for the inviscid flux terms of the governing equations. Central differencing is used for diffusion terms. Baldwin-Lomax eddy viscosity model[17,15] is currently used to model the effects of turbulence on the mean flow. The model is applied at all solid walls of the geometry. In corners between two walls, the contributions to the eddy viscosity from the walls is averaged as shown in Ref. 15. Boundary conditions are implemented using ghost cells. The types of boundary condition to apply at physical boundaries are conveniently specified at runtime.

Solutions are marched to steady state using four stage Runge-Kutta time stepping scheme. Convergence to steady state is accelerated by using local time stepping, implicit residual smoothing and multigrid. To reduce overhead due to communication of data between blocks in multiblock grid systems, exchange of data between blocks takes place only before the first stage of the Runge-Kutta scheme. Also, implicit residual smoothing is done independently within each block.

The TRAF3D.MB code was specifically designed for use with mutliblock grid systems such as the one used in the current study. The code is eqquipped with a completely general multiblock capability, i.e., there is no limitation on the number of blocks a grid system may have or on how they can be connected. Ghost cells are used to accomplish the necessary communication between the blocks of the multiblock grid system. The ghost cells, in effect, create an overlap between blocks and communication between blocks simply involves loading into the appropriate ghost cells the data needed from neighboring blocks. In the current implementation two layers of ghost cells are used around each block of the grid system, creating, in effect, a four-cell wide overlap between blocks. This overlap allows the use of fourth-difference artificial dissipation or up to third order accurate upwind formulas for the inviscid terms of the governing equations without loss of accuracy at blocks boundaries. Communication takes place on all levels of the multigrid cycle at the beginning of each smoothing sweep.

The TRAF3D.MB code has been used in studies of various turbomachinery and prototype flows. Two simulations in particular serve as validations of the accuracy and capability of the flow solver. The first was a simulation of a laminar flow over a backward facing step. Exellent agreement with available experimental data was obtained for both reattachement length and velocity profiles, with reattachment length predicted to within 3% of the experimentally measured values. The results are reported in Ref. 18. The second simulation was of the flow and heat transfer in a turbine rotor cascade, where detailed resolution of the flow field in the tip clearance region was obtained. The predicted values of heat transfer on the rotor tip matched very well with experimentally determined values. The results frome these numerical simulations are described in Ref. 19.

Results

Simulations of flow in the branched duct are currently under way. At the time of printing of this paper, only very preliminary results are available. The results presented here are for flow conditions modeled after the medium Reynolds number case ($Re = 335000$) reported in Ref. 13. The appropriate flow conditions were obtained by specifying, along with the Reynolds number, the static back pressure at the exit of the lower and upper branches of the duct to be the same as in the experiments (pressure specified as a ratio of the inlet total pressure).

The boundary conditions used for the computations presented here are as follows: The static pressure ratios at the exit of the upper and lower branches of the duct were specified as 0.9685 and 0.9725, respectively. At the inlet, a constant total pressure was specified, equivalent to assuming a flat incoming velocity profile. At all solid walls, a no-slip condition was imposed. Wall temperature was fixed at 1.1 times the inlet total temperature. These boundary conditions are only an approximation to the actual run conditions used in the experiments. The most significant difference is that in the experiments, a flow with a developed boundary layer entered the test section. Consequently, the mass flow rate obtained in the simulations is likely in excess of the experimental value.

A sample of results from the simulations is shown in Fig's 5–8. Figure 5 shows the velocity field in the symmetry plane of the duct. Only shown is the region around the pin array and partition, where the topology of the grid is most complicated. Note, velocity at every-other grid point is plotted. Figure 6 shows a close up view of the flow around some of the pins and the leading edge of the partition. Figure 5 reveals how the array of pins acts as a blockage that diverts the flow towards the upper channel of the duct. Consequently the flow approaches the partition at an angle of attack close to 60°. The flow separates above the partition as expected. The blockage effect also causes the flow to enter the pin array with a non-uniform angle. This contributes to a complicated flow field within the pin array (see e.g., Fig. 6) and behind the pins.

Figure 7 shows the velocity field near the expansion corner. As in Fig. 5 and 6, the flow in the symmetry plane of the duct is plotted. Every other vector on grid lines normal to the solid wall is plotted. As the figure shows, the flow separates at the corner. The details of the flow at the corner are well captured without excessive resolution away from the corner.

Finally, Fig. 8 shows contours of static pressure ratio around the leading edge of the partition. Note the continuity of the contours across all block boundaries. As a rule, this continuity is obtained when multiblock grids are used since no interpolations are needed at block boundaries.

Conclusions

Although only preliminary computations of the branched duct flow have been formed thus far, it is still possible to draw important conclusions about the suitability of the present methodology for simulation of viscous flows in complex geometries. A first conclusion is related to the use of multiblock grid systems to model the complicated geometry. A high quality multiblock grid system was successfully generated for the complicated branched duct geometry. The grid obtained was smooth and nearly orthogonal. In particular, high quality grid was obtained around all solid walls. In the 2-D cross section, about 65% of grid points were located in the blocks around the pins, partition and side walls, with the remaining 35% located away from the walls. Thus, one may conclude that other types of structured grids, such as Chimera grids, with similar resolution of the boundary regions, would require about the same total number of grid points as the current grid scheme. On the other hand, the use of complicated interpolation schemes to transfer data between blocks, as is needed when Chimera grids are used, is avoided in the the multiblock flow solver.

In the current approach, the grid system was generated in many small blocks. Although it is possible to use fewer blocks to generate the grid, the fact remains that multiblock grid systems for complicated geometries tend to come in many small blocks. For two reasons, this can be a drawback. First, for flow solvers that run on vector supercomputers, the small blocks tend to lead to short vector lengths and degradation in performance. This effect was observed for the flow solver used in this study. This can be remedied by merging small blocks and form larger ones that allow greater vector lengths, or by writing the computer codes in special ways that allow greater vector lengths. Second, using many small blocks will impact negatively the rate of convergence to steady state if coupling between the blocks is not sufficiently strong. At this time definitive conclusions about the importance of this effect as it relates to the present flowsolver cannot be drawn as the needed numerical experiments have not yet been performed. Indications are, however, this is not a serious effect and that coupling between blocks in the present algorithm is strong enough.

A final observation to be made relates to the use of Baldwin-Lomax turbulence model. Although the Baldwin-Lomax model has been found to work surprisingly well in many complicated flows (see, e.g., Ref. 19) it can not be expected to work perfectly in the current application, due to the complicated 3-D flow field, for instance behind the array of pins in the lower branch of the duct. For future detailed simulations, a more general two-equation turbulence model should be used.

Acknowledgements

The first author wants to thank Douglas R. Thurman for providing data from experiments on the branched duct, and for giving insights into the nature of the flow in the duct.

References

1 Thompson, J.F., Thames, F.C., and Mastin, C.W., "Automatic Numerical Grid Generation of Body-Fitted Curvi-linear Coordinate System for Field Containing Any Number of Arbitrary Two-Dimensional Bpodies", *J. of Computational Physics*, Vol. 15, pp. 299-319, 1974.

2 Steger, J.L., Dougherty, F.C. and Benec, J.A., "A Chimera Grid Scheme," Advances in Grid Generation, Ed. by K.N. Ghia, and U. Ghia, 1983, pp. 59-70.

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

4 Benek, J.A., Steger, J.L., Dougherty, F.C., and Buning, P.G., "Chimera: A Grid-Embedding Technique," AEDC-TR-85-64, April 1986.

5 Rimlinger, M.J., Shih, T. I-P., and Chyu, W.J., "Three-Dimensional Shock-Wave / Boundary-Layer Interactions with Bleed Through a Circular Hole," AIAA-92-3084.

6 Kao, K.-H., Liou, M.-S. and Chow, C.-Y., "Grid Adaption Using Chimera Composite Overlapping Meshes," AIAA-93-3389, Also; NASA TM 106163, and ICOMP-93-16.

7 Shus, N.E. and Tramel, R.W., "PEGASUS 4.0 User's Manual," AEDC-TR-91-8, November 1991.

8 Allwright, S. E., "Techniques in Multiblock Domain Decomposition and Surface Grid Generation," in *Numerical Grid Generation for Computational Fluid Mechanics '88*; Sengupta, S., Häuser, J., Eiseman, P. R., and Thompson, J. F., (ed), pp. 559-568, Pineridge Press, 1988.

9 Amdahl, D. J., "Interactive Multi-Block Grid Generation," in *Numerical Grid Generation for Computational Fluid Mechanics '88*; Sengupta, S., Häuser, J., Eiseman, P. R., and Thompson, J. F., (ed), pp. 579-578, Pineridge Press, 1988.

10 Stewart, M., "A General Decomposition Algorithm Applied to Multi-Element Airfoil Grids," AIAA 90-1606, 1990.

11 "GRIDPRO,^{TM} /AZ3000, Users Guide and Reference Manual", Program Development Corporation, White Plains, NY, 1993.

12 Schönfeld, T. and Weinerfelt, P., "Automatic Generation of Quadrilateral Multi-Block Grids by the Advancing Front Technique," Proceedings of the *Third International Conference on Numerical Grid Generation*, pp. 743-754, S.-Arcilla, A., Häuser, J., Eisemann, P.R., and Thompson, J.F. (Eds.), Elsevier Science Publishers, North Holland, June 1991.

13 Russel, L.M., Hippensteele, S.A., Poinsatte, P.E., 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-93-1797, 1993. Also; NASA TM 106189.

14 Steinthorsson, E., Liou, M.-S., and Povinelli, L.A., "Development of an Explicit Multiblock / Multigrid Flow Solver for Viscous Flows in Complex Geometries," AIAA-93-2380, 1993. Also; NASA TM 106356 and ICOMP 93-34.

15 Arnone, A., Liou, M.-S., and Povinelli, L. A., "Multigrid Calculation of Three-Dimensional Viscous Cascade Flows," AIAA-91-3238, Sept. 1991.

16 Liou, M.-S., and Steffen, C. J., "A New Flux Splitting Scheme," NASA TM 104404, May 1991.

17 Baldwin, B. S., and Lomax, H., "Thin Layer Approximation and Algebraic Model for Separated Turbulent Flows," AIAA paper 78-257, 1978.

18 Steinthorsson, E., Liou, M.-S., Povinelli, L. A. and Arnone, A., "Numerical Simulations of Three- Dimensional Laminar Flow Over A Backward Facing Step; Flow Near Side Walls,' 'FED Vol 149, pp. 19-26, ASME, 1993.

19 Ameri, A., and Steinthorsson, E., "Prediction of Unshrouded Rotor Blade Tip Heat Transfer," to be presented at ASME Int. Gas Turbine Conference and Exposition, Houston, Texas, June, 1995.

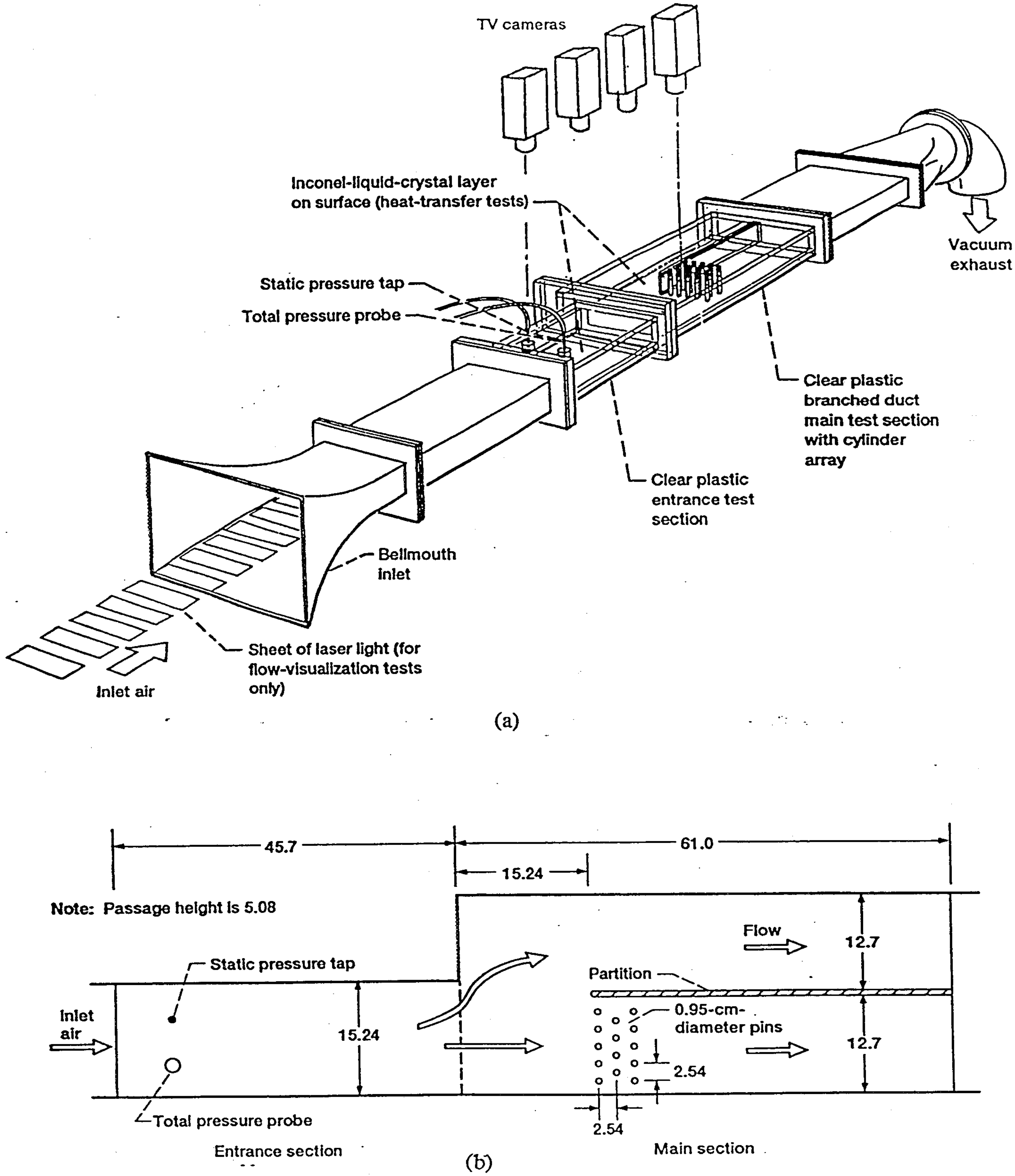
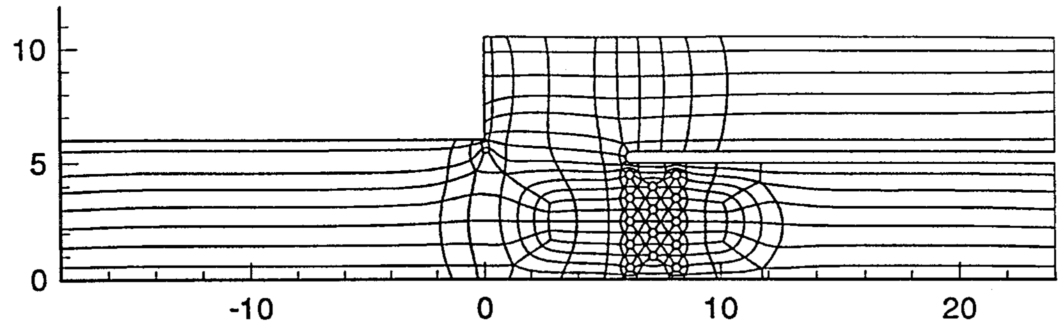
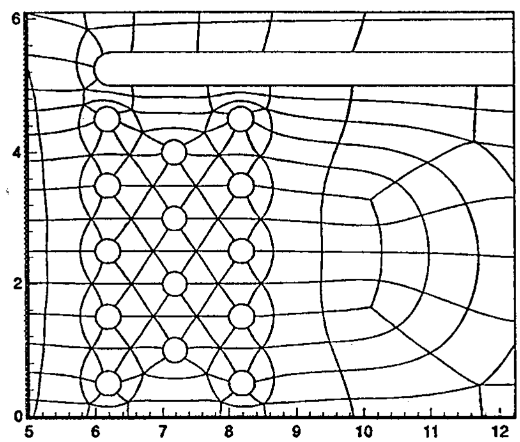


Figure 1. Branched duct geometry (a) experimental setup (b) schematic drawing of the test section (all dimensions are in centimeters). Figure reprinted from Ref. 13 (with permission).



(a)



(b)

Figure 2. Topology of multiblock grid system (a) overall grid, (b) around pin array and partition

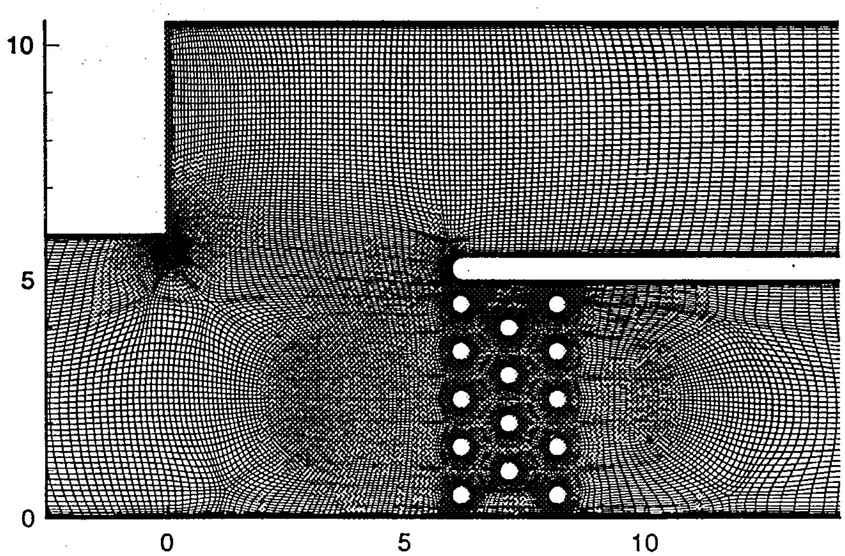
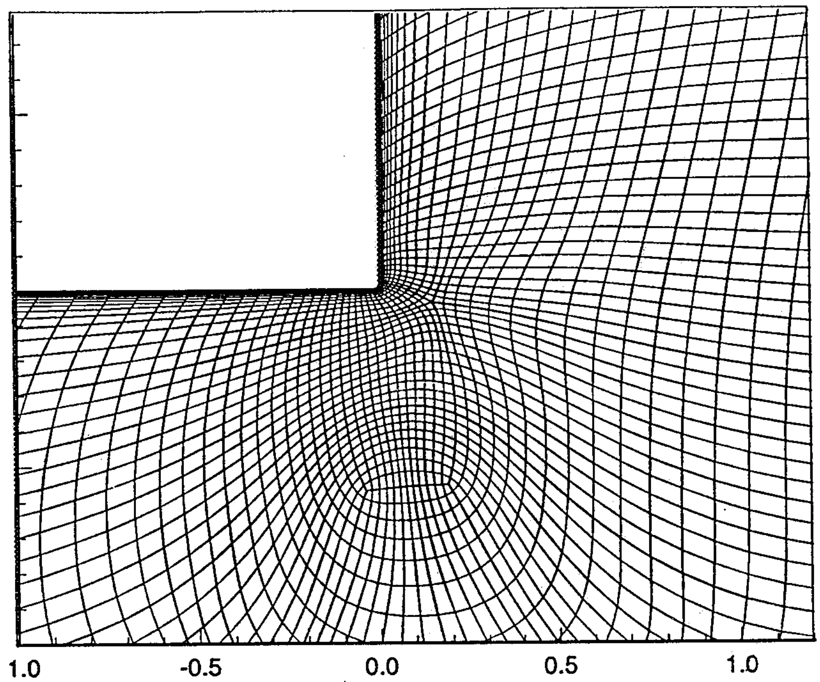
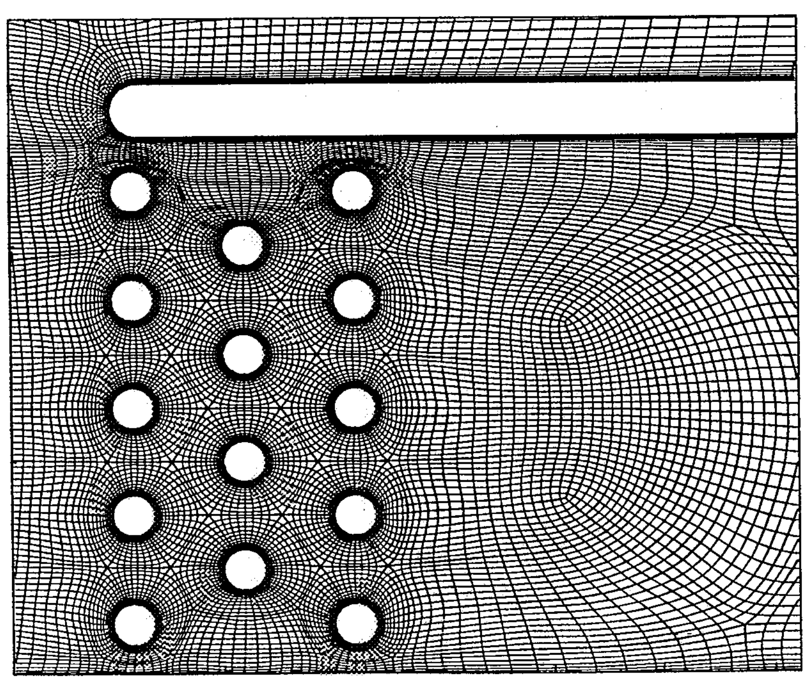


Figure 3. Grid system for the center region of the branched duct.



(a)



(b)

Figure 4. Close-up view of grid near (a) expansion corner and (b) pin array and position

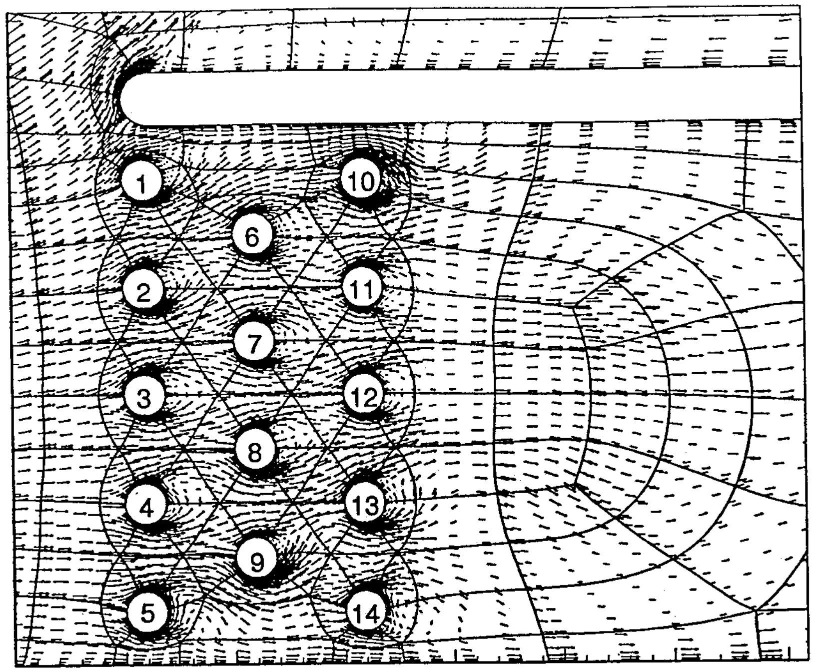


Figure 5. Velocity field in the lower branch (symmetry plane)

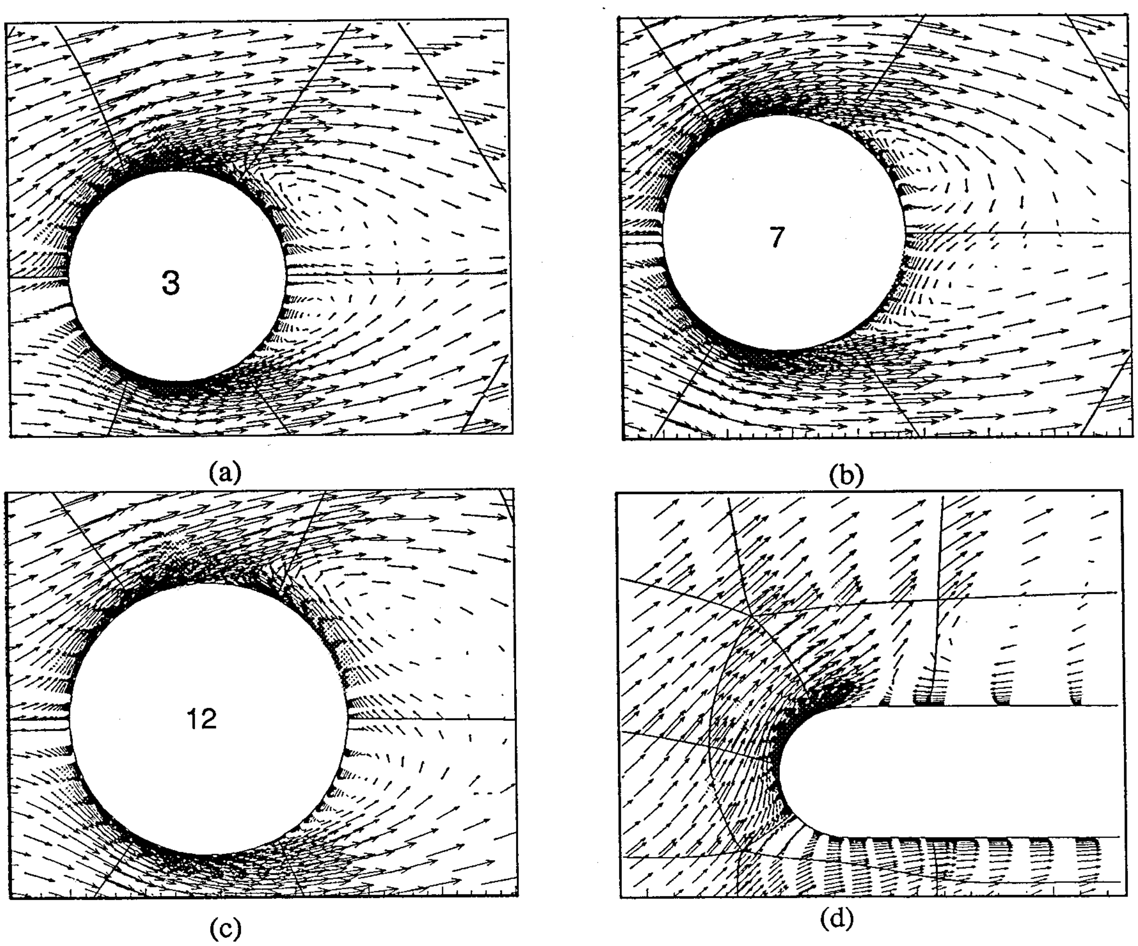


Figure 6. Close-up view of velocity field near (a) pin 3, (b) pin 7, (c) pin 12 and (d) leading edge of the partition.

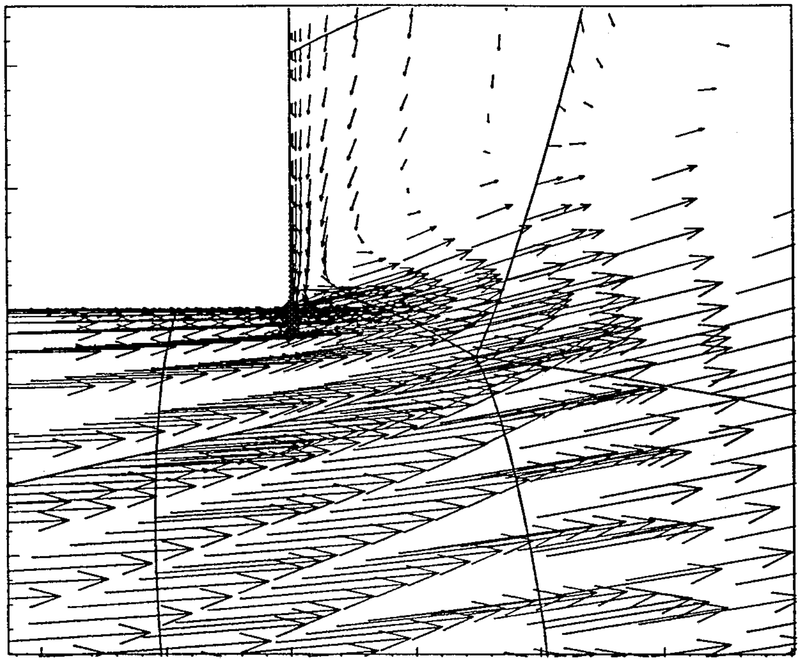


Figure 7. Velocity field near expansion corner (symmetry plane)

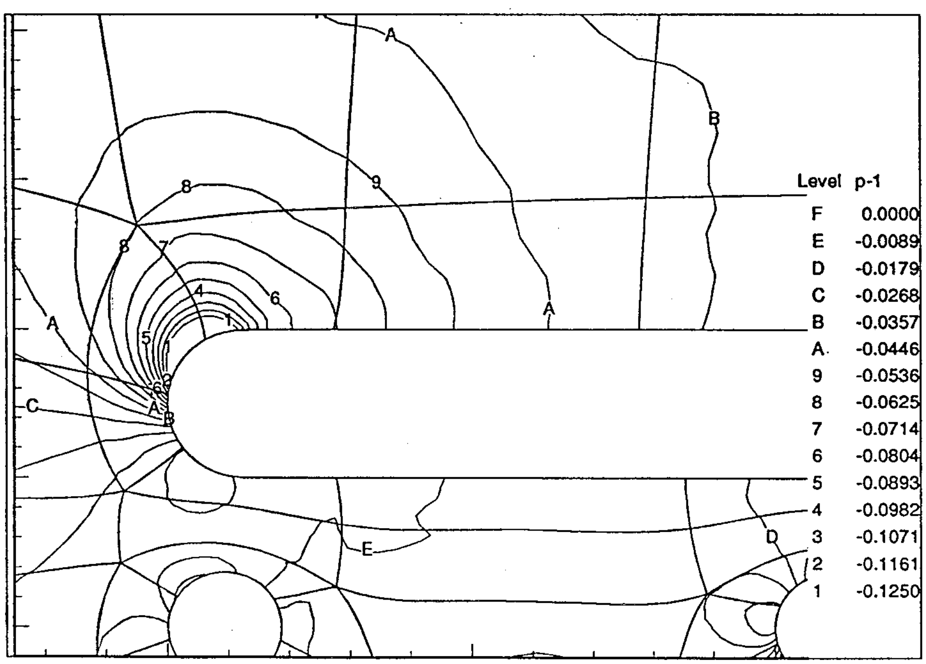


Figure 8. Static pressure ratio contours near leading edge of the partition (symmetry plane).

REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

1. AGENCY USE ONLY (*Leave blank*)	2. REPORT DATE	3. REPORT TYPE AND DATES COVERED
	November 1995	Contractor Report

4. TITLE AND SUBTITLE
Computations of Viscous Flows in Complex Geometries Using Multiblock Grid Systems

5. FUNDING NUMBERS
WU–505–90–5K

6. AUTHOR(S)
Erlendur Steinthorsson and Ali A. Ameri

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)
Institute for Computational Mechanics in Propulsion
22800 Cedar Point Road
Cleveland, Ohio 44142

8. PERFORMING ORGANIZATION REPORT NUMBER
E–9992

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)
National Aeronautics and Space Administration
Washington, D.C. 20546–0001

10. SPONSORING/MONITORING AGENCY REPORT NUMBER
NASA CR–198419
ICOMP 95–23
AIAA–95–0177

11. SUPPLEMENTARY NOTES
Prepared for the 33rd Aerospace Sciences Meeting and Exhibit sponsored by the American Institute of Aeronautics and Astronautics, Reno, Nevada, January 9–12, 1995. Erlendur Steinthorsson, Institute for Computational Mechanics in Propulsion, NASA Lewis Research Center, work funded under NASA Cooperative Agreement NCC3–370); Ali A. Ameri, AYT Corporation, Beachwood, Ohio. ICOMP Program Director, Louis A. Povinelli, organization code 2600, (216) 433–5818.

12a. DISTRIBUTION/AVAILABILITY STATEMENT
Unclassified - Unlimited
Subject Categories 34 and 64

This publication is available from the NASA Center for Aerospace Information, (301) 621–0390.

12b. DISTRIBUTION CODE

13. ABSTRACT *(Maximum 200 words)*

Generating high quality, structured, continuous, body-fitted grid systems (multiblock grid systems) for complicated geometries has long been a most labor-intensive and frustrating part of simulating flows in complicated geometries. Recently, new methodologies and software have emerged that greatly reduce the human effort required to generate high quality multiblock grid systems for complicated geometries. These methods and software require minimal input from the user–typically, only information about the topology of the block structure and number of grid points. This paper demonstrates the use of the new breed of multiblock grid systems in simulations of internal flows in complicated geometries. The geometry used in this study is a duct with a sudden expansion, a partition and an array of cylindrical pins. This geometry has many of the features typical of internal coolant passages in turbine blades. The grid system used in this study was generated using a commercially available grid generator. The simulations were done using a recently developed flow solver, TRAF3D.MB, that was specially designed to use multiblock grid systems.

14. SUBJECT TERMS
CFD; Multigrid; Multiblock grid; Turbine cooling

15. NUMBER OF PAGES
12

16. PRICE CODE
A03

17. SECURITY CLASSIFICATION OF REPORT	18. SECURITY CLASSIFICATION OF THIS PAGE	19. SECURITY CLASSIFICATION OF ABSTRACT	20. LIMITATION OF ABSTRACT
Unclassified	Unclassified	Unclassified	