1995122341

N95-28762

AUTOMATION OF THREE-DIMENSIONAL STRUCTURED MESH GENERATION FOR TURBOMACHINERY BLADE PASSAGES

Edward P. Ascoli and George H. Prueger
Rockwell International/Rocketdyne Division
6633 Canoga Avenue, P.O. Box 7922
Canoga Park, CA 91309-7922

ABSTRACT

Hybrid tools have been developed which greatly reduce the time required to generate three-dimensional structured CFD meshes for turbomachinery blade passages. RAGGS, an existing, Rockwell proprietary, general purpose mesh generation and visualization system provides the starting point and framework for tool development. Utilities which manipulate and interface with RAGGS tools have been developed to 1) facilitate blade geometry inputs from point or CAD representations, 2) automate auxiliary surface creation and 3) streamline and automate edge, surface and subsequent volume mesh generation from minimal inputs. The emphasis of this approach has been to maintain all the functionality of the general purpose mesh generator while simultaneously eliminating the bulk of the repetitive and tedious manual steps in the mesh generation process. Using this approach, mesh generation cycle times have been reduced from the order of days down to the order of hours.

INTRODUCTION

Over the last decade, advances in both computer technology and numerical algorithms have made Computational Fluid Dynamics (CFD) a viable analysis tool for engineering design. In particular, aerospace companies have employed CFD in the design of turbomachinery components, combustion devices as well as hypersonic propulsion systems. The major limitation to CFD as a design tool has been the length of the CFD analysis process. When modeling complex geometries the CFD analysis process is frequently too time intensive to be used early in the design cycle. Unfortunately, the later that CFD is used in the design cycle the less likely it is to have an impact on the final design.

Geometrical complexity impacts the CFD analysis process at each step. In the preprocessing step, the time required for model creation and mesh generation increases dramatically with increasing geometrical complexity. CFD meshes for complex geometries generally require more nodes or elements and thus the flow solution phase of the CFD process increases in time. Finally, visualization and data reduction in the postprocessing phase require more time and thought for complex geometries. On average, for complex geometries, the preprocessing phase of the analysis requires the greatest amount of time. In many cases preprocessing may require much more time than either flow solution or postprocessing combined. The primary focus of this work is on the preprocessing step of the CFD analysis cycle.

Preprocessing logically subdivides into model creation and mesh generation. The geometry may only exist in the form of diagrams or sets of points defining lines, curves or surfaces. Tools are then required to convert this data form into a model upon which a mesh may be constructed. In this case, a model may be created using any of a variety of CAD tools such as CATIA or Pro/ENGINEER. Mesh generation tools would then be required to import the CAD geometry. Alternately, mesh generation packages exist which have CAD or CAD-like capability.

A CAD model may have already been created to serve some other design purpose. Unfortunately, a CAD model suitable for rapid prototyping or automated machining etc. may be different than a model of the same geometry which is suitable for mesh generation. Extraneous surfaces may have to be removed while additional (auxiliary) surfaces may have to be created to extend or partition a flow region.

Once a suitable model has been obtained, mesh generation is performed. Although fully unstructured mesh CFD technology has been rapidly evolving, it has not reached the maturity to displace existing multiblock structured mesh CFD technology. This work focuses on structured mesh generation. Structured mesh generation requires up front planning to decide the general topological layout and blocking, as well as node number and distribution. In general, three-dimensional structured meshing proceeds serially from edge meshing to surface meshing and then to volume meshing. Four edges determine the boundary of a non-degenerate surface while six surfaces determine the boundary of a non-degenerate volume.

The goal of this effort has been to reduce the time spent in the preprocessing phase of the CFD analysis cycle for a commercially important class of complex geometries. To this end 1) a relevant class of geometries was selected, 2) the primary grid generation tool was identified, 3) key geometry model input and generation issues were identified, 4) key flow solution requirements were identified and 5) tools were developed which automated and expedited the preprocessing phase of the analysis cycle.

GEOMETRY CLASS

A relevant class of complex geometries, namely blade passages, was identified. Blade passages are elements in a wide variety of applications ranging from rocket engine turbopumps (ref. 1) to air compressors. The primary deterrent to using full three-dimensional CFD in the initial design of rocket engine turbopumps is the time required to generate the appropriate geometry model and corresponding mesh. In this effort, three types of blade passages were considered: axial, radial and mixed. Axial flow devices have inflow and outflow primarily in the axial direction. Radial flow devices have axial inflow and radial outflow. Mixed flow devices have inflow in the axial direction and outflow at a flow angle between axial and radial. The three configurations are shown schematically in figure 1. Figures 2 through 4 are turbopump components with either axial or radial flow configurations.

PRIMARY GRID GENERATION SYSTEM

Rockwell Automated Grid Generation System (RAGGS, (ref. 2)), a general purpose mesh generation and visualization system was selected as the primary grid generation tool. RAGGS is a Rockwell proprietary code which was developed at North American Aircraft Division of Rockwell International. In interactive mode, RAGGS is a highly graphical tool with an extensive user interface. Geometry models as well as edge, surface and volume meshes are easily manipulated and visualized via menus and mouse control. Edge, surface and volume meshing may be performed in a highly interactive and visual manner in this mode as well. Of critical importance to the current effort is the fact that nearly every primary gridding tool available in interactive mode has a nongraphical, "batch" counterpart. For instance, a simple nongraphical tool exists to create an edge mesh on a specified surface. Similarly, nongraphical tools exist for surface and volume meshing. These tools are extremely accessible and flexible. The key point is that these tools may be combined to form macros. Since RAGGS runs in a UNIX workstation environment, C-shell and Bourne shell macros were written calling various tools. In this manner the grid generation process was expedited and automated while at the same time key intermediates were easily visualized and optimized in interactive mode.

GEOMETRY INPUT AND GENERATION

Initial geometry data for blade passage configurations at Rocketdyne is available in one of two forms: point data or CAD geometry files. Generally, point data is available earlier in the design cycle. The format for point data is two arrays of points, one for each side of the blade. Together the two arrays may combine to form the two sides of a single blade or the two arrays may represent the bounding blade surfaces of a passage. The arrays are assumed to be N x M arrays (M curves with N points per curve) which span the leading edge to trailing edge, otherwise points are interpolated as appropriate. From the blade surface data and a few geometrical inputs (e.g., the number of blade passages) all other geometrical features can be constructed.

Since CAD design at Rocketdyne may be performed on a variety of CAD systems, CAD geometry files exist in any of the corresponding CAD formats. All of the CAD systems at Rocketdyne have translators enabling conversion to IGES (International Graphics Exchange Standard) format. RAGGS is able to input IGES geometry files and the CAD to IGES to RAGGS path has been tested for CATIA, Pro/ENGINEER, PATRAN, and UNIGRAPHICS. CAD geometry files obtained at this stage in the design cycle usually contain a large amount of superfluous detail (i.e. details of solid portions of the hardware item when flow passages are of primary interest, figure 5). Often the geometry may not contain a complete contiguous blade passage (i.e., the furnished geometry may consist of a "pie" or sector cut of the full geometry, figure 6). For this effort, when CAD data rather than point data was provided it was assumed that the format was IGES and that two complete blade surfaces could be constructed from the IGES data provided. Again, the two surfaces could combine to form either a single blade or the bounding surfaces of a passage.

FLOW SOLUTION CONSIDERATIONS

Simulation of the flow domain requires that the blade passage be extended both upstream and downstream. The length of the upstream and downstream extensions is an input provided by the analyst performing the flow simulation. Although the geometry details of the upstream and downstream extensions are flexible, the side boundary surface grids must be periodic since the single blade passage along with the extensions represents one element in a device with repeated elements in rotational symmetry.

The flow algorithm and physical modeling also dictate a host of grid features including wall clustering, cell aspect ratio, cell skewness and overall grid quality.

TOOL DEVELOPMENT

The preprocessing phase of the analysis cycle was divided into the following steps: 1) blade surface input, 2) full geometry creation, 3) edge and surface meshing, 4) volume meshing and 5) mesh manipulation and quality checking. Corresponding to each step, a module was created.

The blade surface module converted either IGES blade geometry data or blade point data into a form which could be directly used by RAGGS. The output of the blade surface module could be read directly into RAGGS in interactive mode and examined for consistency and correctness.

The geometry module read user geometry input (e.g., upstream and downstream extension lengths and geometry details) as well as the blade surface geometry as output from the blade surface module. The geometry module used the inner and outer edges of the blade surface to construct the hub and shroud profiles. The blade leading and trailing edges were used to construct the upstream and downstream extensions respectively. The blade surfaces could be given as either the two sides of a single complete blade or as two sides of the blade passage. If the surfaces were given as a single complete blade the logic was in place to generate a blade passage. Based on geometrical input, two auxiliary surfaces were created, one spanned the entrance of the blade passage from leading edge to leading edge while the second spanned the exit of the blade passage from trailing edge to trailing edge. These two auxiliary surfaces partitioned the total geometry into upstream extension, blade passage and downstream extension. Partitioning the geometry in this way provided greater control over the three volume grids that were subsequently generated. In particular, these auxiliary surfaces were used to minimize mesh skewness at the blade leading and trailing edges.

The geometry module centered about FORTRAN programs written to create the new geometry elements. These FORTRAN programs along with existing batch RAGGS tools were linked via C-shell scripts. The output of the geometry module was a complete consistent geometry containing all features necessary to make the final mesh. This geometry could be read into RAGGS in interactive mode and examined.

The edge and surface meshing module read point distribution and clustering information from an input deck and automatically generated the appropriate edge and surface meshes. Initial surface meshes were generated with RAGGS transfinite mesh generation tools. For surfaces which were anticipated to contain excessively skewed cells, elliptic refinement was automatically performed. Each surface mesh was tested for negative area cells. If negative area cells were encountered, elliptic refinement was performed automatically. The output of the edge and surface meshing module was a set of files containing final edge and surface meshes. These files were easily displayed and modified in interactive RAGGS mode.

The volume mesh module read surface meshes from the edge and surface meshing module and produced transfinite volume grids. Each volume grid was tested for negative cell volumes. If a negative cell volume was encountered elliptic refinement was performed automatically. The final volume mesh was examined in RAGGS interactive mode.

The last module performed a host of miscellaneous manipulations and quality checks on volume meshes. Minimum, maximum, average and standard deviation were calculated for quantities like aspect ratio, skewness and cell volume. Grid index manipulations could be performed as needed. Thus the I index could be switched with the K index, etc. Two or more grids could be combined into a single grid. Since the blade passage grid was actually generated in three blocks corresponding to the inflow extension, the blade passage and the outflow extension, if the analyst preferred a single block, the three blocks were combined into one block with this tool. Mesh lines could be added (interpolated) or removed using the mesh manipulation tool. Finally, the mesh could be output in any of a variety of formats required by existing flow solvers.

RESULTS

Figure 7 shows the three-dimensional mesh generated for the axial configuration modeling a three-dimensional turbine nozzle. The blade surface for this configuration was available both as blade point data as well as CAD IGES data. Both formats were meshed using the tools developed under this effort. Beginning with either blade point data or CAD IGES geometry data, the start to finish meshing time was less than five hours using the automated tools. The five hours roughly divides into three hours for geometry creation and finalization and less than two hours for actual mesh generation. The total of five hours compares to 25 hours for "manual" meshing from an existing CAD IGES geometry and 30 hours for "manual" meshing beginning from blade point data. When the geometry is available as blade point data, manual refers to, first, employing the services of a CAD expert to create the base geometry and generate the necessary auxiliary surfaces to partition the flow passage and, second, using RAGGS in interactive graphical mode to generate the mesh step by step. When the geometry is available as a CAD IGES file the first manual step is simplified since the CAD expert need not generate the base geometry.

The axial flow devices shown in figures 8 and 9 are both inducers. Blade point data was provided in each case. Start to finish mesh generation using the automated tools required between five and six hours for each configuration. This time compares to roughly 35 hours for "manual" (RAGGS and CAD expert) mesh generation. Note that it took two hours to produce a subsequent modified mesh for the configuration shown in figure 9. This modified mesh employed the same geometry as shown in the figure, but varied point distribution and clustering.

Figure 10 shows an impeller. Blade data for this radial flow device was provided in the form of a CAD IGES file. Time estimates for "manual" and "automated" generation of this configuration are nearly the same as those given above for the inducer configurations.

FUTURE WORK

Future work will focus on including a variety of additional geometrical features. The overall approach can be modified to include partial or splitter blades in the blade passage. Tools exist to handle tip clearances between the blade and the shroud but these tools have not been incorporated into the current framework. Subsequent work will improve these tools and incorporated them into the overall system. Fillets at the intersection of blade surface with the hub or shroud could eventually be taken into account. Currently, blade surface IGES files which are based on trimmed surfaces are excluded. Future work will eliminate this limitation. Finally, this work has focused on rotating machinery blade passages. The approach that has been taken is sufficiently general that it can be applied to nearly any geometric class to produce template type capability. Future work will focus on other geometric classes to produce preprocessing templates analogous to the one presented in this paper.

REFERENCES

1. AIAA-92-3400, "Application of a Computational Engineering System to Analysis of a Rocket Engine Pump Impeller," Steven L. Barson, Edward P. Ascoli, Daniel C. Chan, Michele E. DeCroix, and Munir M. Sindir, 28th Joint Propulsion Conference and Exhibit, July 6-8, 1992, Nashville, TN.

2. AIAA 94-0206, "RAGGS: Rockwell Automated Grid Generation System," Chung-Jin Woan, Willard C. Clever and Clement K. Tam, 32nd Aerospace Science Meeting, Jan. 10-13, 1994, Reno Nevada.

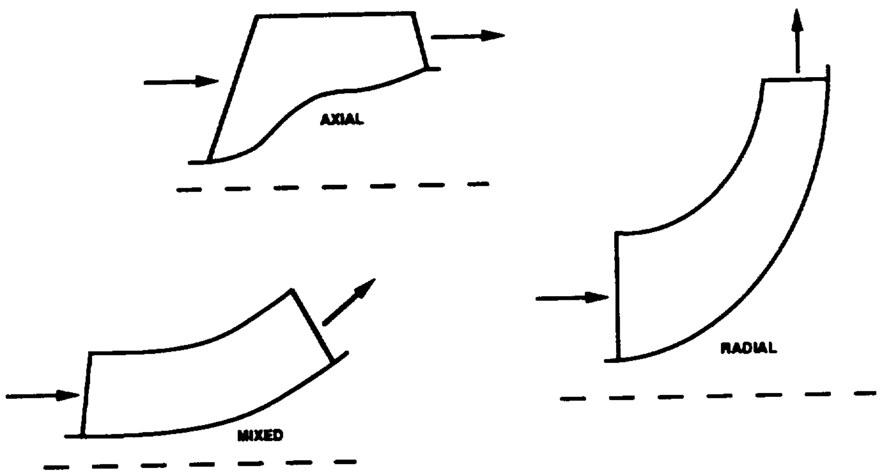

Figure 1. –Schematic of axial, radial and mixed blade passages.

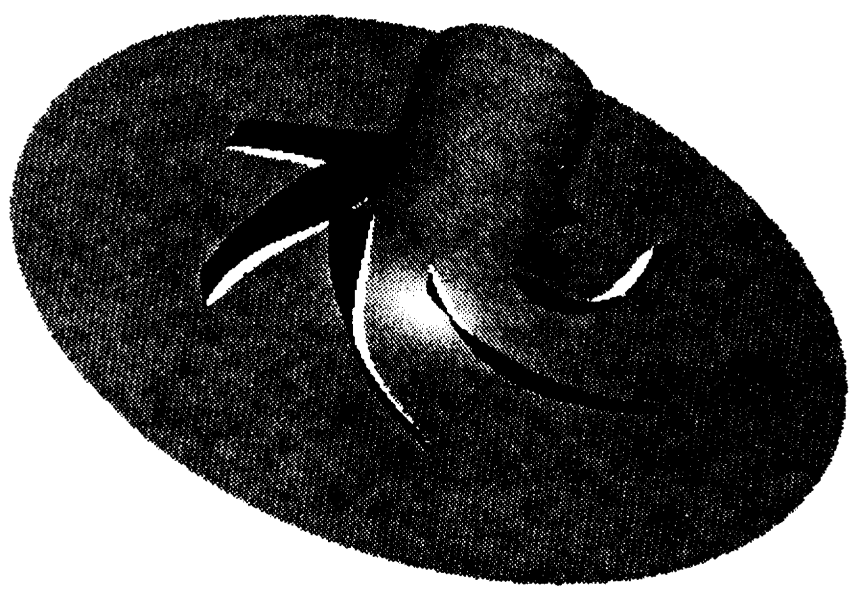

Figure 2.–Radial flow device (Impeller).

Figure 3.–Axial flow device (three-dimensional turbine nozzle)

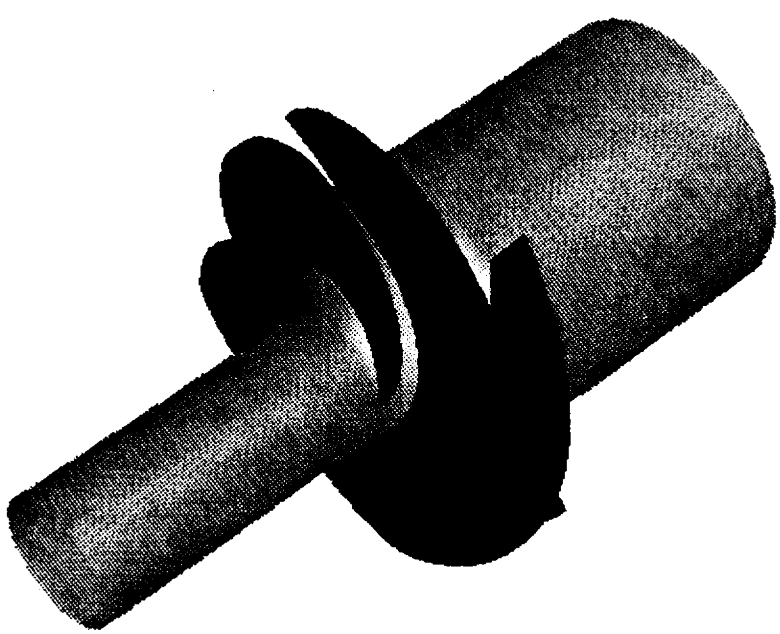

Figure 4.–Axial flow device (inducer)

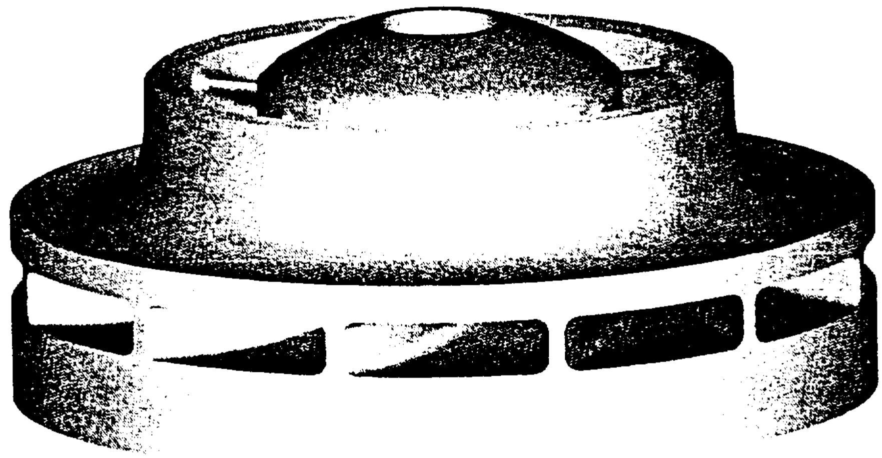

Figure 5.–CAD geometry representation of an impeller showing full 360° display and superfluous hardware detail.

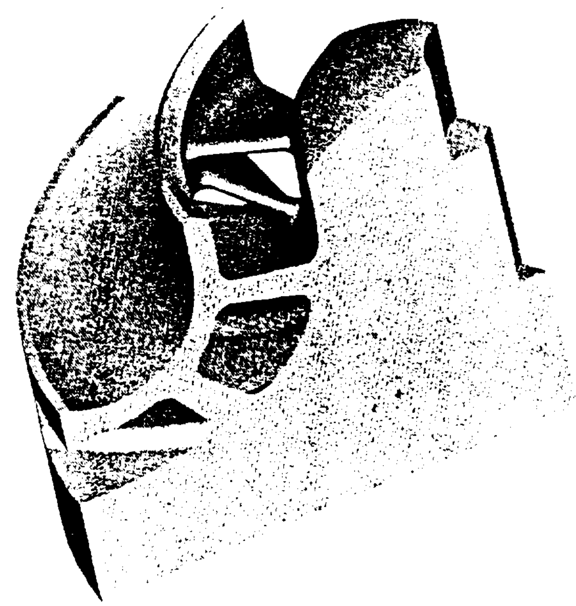

Figure 6.–A "pie-slice" or sector cut of the geometry shown in figure 5.

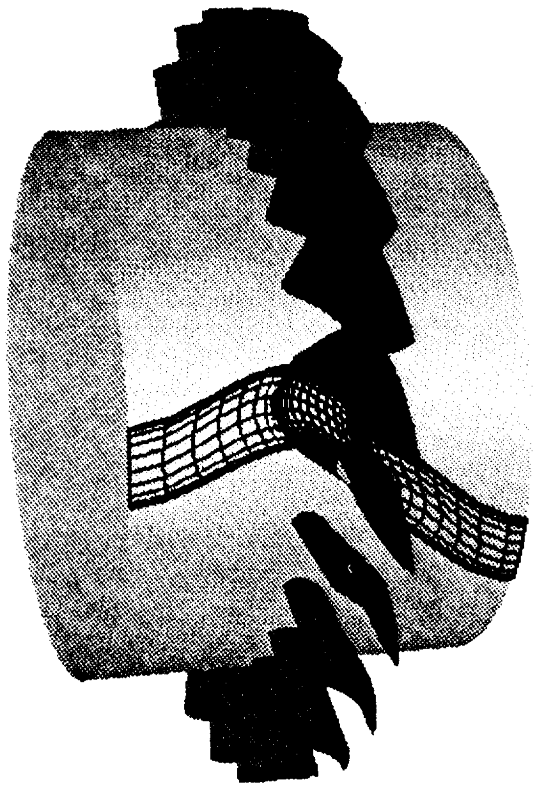

Figure 7.–Representative mesh surfaces for a three-dimensional turbine nozzle (Axial flow device).

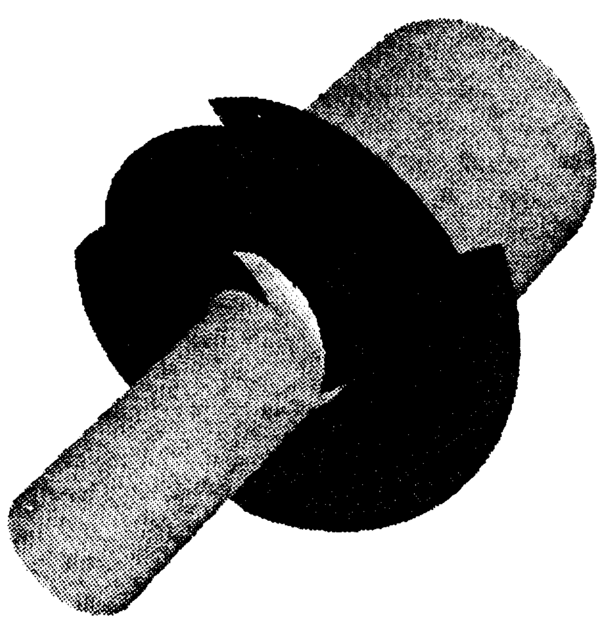

Figure 8.–Representative mesh surface for an inducer (Axial flow device).

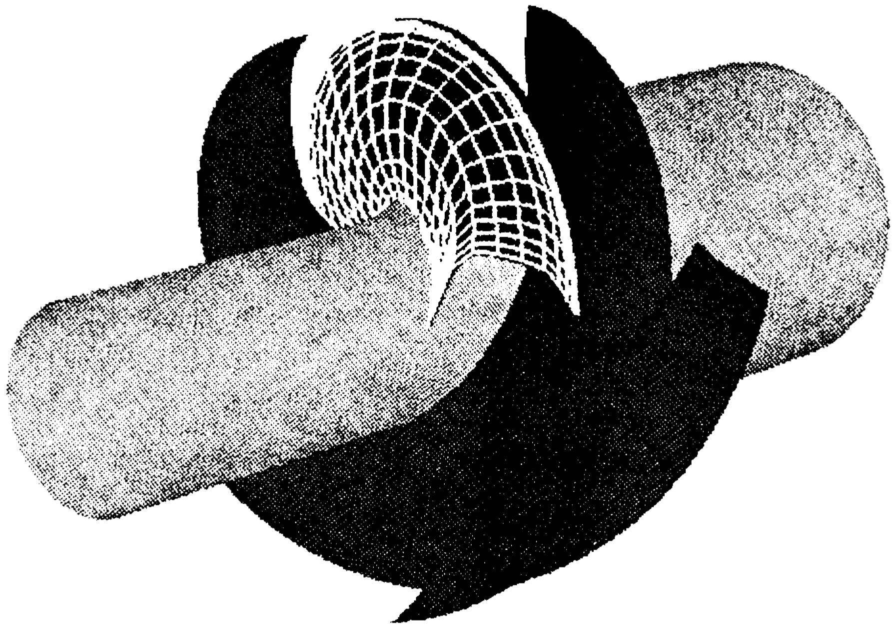

Figure 9.–Representative mesh surface for an inducer with a novel, sickle shaped blade leading edge (Axial flow device).

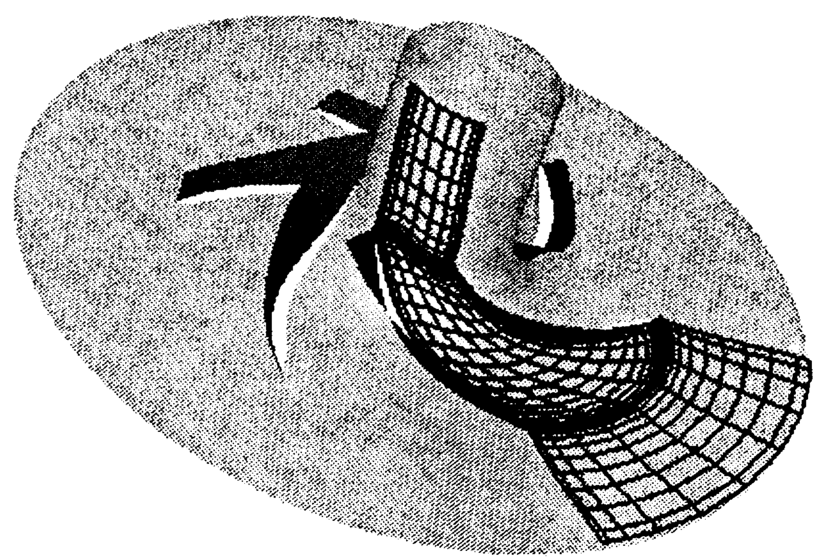

Figure 10.–Representative mesh surfaces for an impeller (Radial flow device).

C-8