Introduction

In this paper, we describe a new class of adaptive finite element methods that have been designed for use in flow interaction problems - particularly interaction problems of the type encountered in flow around rotor-stator configurations in turbomachinery. Four major algorithms are described in this work:

1. A mesh refinement strategy that provides for automatic refinement of a moving finite element mesh when local errors exceed preassigned limits.

2. A mesh unrefinement strategy, that increases element sizes and reduces the number of elements when the local element errors fall below a prescribed level.

3. A mesh distortion/node redistribution algorithm which provides for a repositioning of nodes in a mesh with a fixed number of cells and nodes so as to optimally equidistribute the element errors.

4. A moving mesh algorithm, in which a mesh surrounding one flow domain is moved relative to a mesh surrounding another flow domain along a smooth mesh interface.

The finite element methods which provide a basis for these algorithms are a) a two-step Taylor-Galerkin/Lax-Wendroff explicit scheme based on \( Q_1 \) - bilinear elements with Lapidus viscosity for transient Euler equations in two-dimensions and b) a special PAGE finite element algorithm that uses an explicit MacCormick method. Some calculations have also been performed using the streamline-upwinded Petrov-Galerkin method.

A number of example problems have been solved using these algorithms. Preliminary tests of the algorithms have focused on two-dimensional supersonic flow (Euler equations), a class of two-dimensional convection-diffusion problems, and a moving grid problem for rotor-stator interaction in a two-dimensional flow model. It is planned that the methods will soon be applied to a class of flow interaction problems encountered in the Space Shuttle main engine, high-pressure pumps, where it is important to simulate the fluctuating pressures and dynamic loads due to fluid dynamic interaction between moving and stationary blades and vanes.

Sample numerical results:

We shall outline briefly some typical results of numerical experiments.

1. Flow over a step. A computed refined mesh for the problem of supersonic flow through a wind tunnel with a step introduced in the flow is shown in
The initial uniform mesh contained elements smaller than the largest seen in the figure; thus, both mesh refinement and unrefinement took place to obtain this solution. The same problem was solved using the node redistribution method for a fixed number of elements, and results are shown in Fig. 2. Plots of computed density contours are also shown. Note that both methods attempt to capture the shock and its reflection on the tunnel walls.

2. Supersonic flow in turbomachinery. Results of a first attempt at modeling the flow-interaction of a series of airfoils moving relative to a stationary airfoil are shown in Fig. 3. The rotor mesh slides at a prescribed rate relative to the fixed stator mesh along the straight interface. The evolution of fluid pressures on the rotor blades is computed over 3000 time steps. Computed density profiles for a fixed time are shown. These results are obtained with a very coarse mesh. Further studies are underway which will combine the adaptive procedures and the moving mesh procedures so as to attempt to obtain an optimal mesh at each time step for flow interaction problems.

Acknowledgement. This work has been supported by the National Aeronautics and Space Administration (NASA) through Contract NAS8-36647 and Contracts NAS1-17894 and NAS8-36284 with the Lockheed Missiles and Space Co., Huntsville Division.
Figure 3a.