Pakistan Journal of Applied Sciences 2(10):955-964, 2002

Rotor-Stator Interaction in Turbomachines

[1]E. Shirani and [2]K. Daneshkhah
[1]Isfahan University of Technology, Isfahan, Iran
[2]Kalgery University, Kalgery, Canada

Abstract: Rotor-stator interaction in turbomachines is considered. A computer program was developed to numerically simulate 2-D unsteady compressible Navier-Stokes equations based on Jameson's numerical method. In this problem, the sliding mesh method is used at the interface of rotor and stator. The transfer of information at the sliding interface was done based on a conservative form of a third order interpolation technique. Several test cases were carried out to examine the problem and obtain the effect of rotor-stator interaction. The results show that for the turbine type stages, the stator-rotor interaction affects mainly the moving blades and also the wake region of the stationary blades. The amount of these effects depends on the velocity of the rotor blades.

Key Words: Rotor-stator Interaction, Jameson Method, Turbomachines, Unsteady Flows, Sliding Mesh Method, Turbine Stage

Introduction

One of the most important problems in rigorous design and analysis of modern turbomachines is to obtain the details of the flow field by means of CFD. Many aspects of the flow such as heat transfer, secondary flows and radial mixing can be predicted very well by using viscous flow analysis and the results can be used for industrial applications as well as for design purposes.

There have been several investigations for flows in the blade cascades. The numerical investigations used in this regard, are the solution of two-dimensional flow equations for inviscid and viscous flows. The analysis of such flows was started from 1950's by the solution of two-dimensional incompressible potential flows based on the singular flows and panel method. Some of the main works done for simulation of flow in cascades are as follows. McDonald (1971) and Denton (1975) used Lax-Wendroff method to simulate compressible ideal flows. Crawford and Kays (1976) and Lakshminarayana and Govindan (1981) analyzed two-dimensional flows using boundary layer equations. Stager, *et al.*, (1980), Shamroth *et al.*, (1982) and Knight and Choi (1989) simulated inviscid and viscous flows using Beam-Warming method. Jameson *et al.*, (1981) integrated the governing equations over a grid size control volume and simulated inviscid flow in a cascade. This method which, is called Jameson's method, is also used in this work. Li *et al.*, (1989), Shirani and Zirak (2001) and Arnone and Swanson (1993) used Jameson method to simulate flow in the cascades. Barnett *et al.*, (1990) solved transonic flows in a compressible cascade. Jameson *et al.*, (1981) and Jameson (1991) simulated the compressible flow in a cascade by integration of the equation of motion over a control volume.

Flow in the turbomachines is fully unsteady and is very much affected by pressure waves and wakes in the region between rotor and stator blades. Although the study of the flow in a single blade row, cascade, is useful to consider many fluid dynamics phenomena, but it does not give any information about the unsteady behavior of the flow due to the rotor-stator interaction. When the axial distance between the two blade rows are reduced, the interaction effects are pronounced. The results obtained by Dering *et al.*, (1982) show that when the axial distance between the two blade rows is 15% of the blade length, the difference between the maximum and minimum pressures near the tip of the rotor blades, in one period, is about 70% of the exit dynamic pressure. Such a small axial distance between the two blade rows is very common in the modern turbomachines. So it is necessary to consider rotor and stator as one system and study their interactions.

Rotor-stator interaction is the main parameter for unsteadiness of the flow in turbomachines. It introduces some changes in the turbomachine performance. For the same reason, it attracts some researchers. Some of the main researches done in this area are as follows. Erdos *et al.*, (1972) simulated the flow in an axial fan stage by using McCormack method. They used unequal pitch for rotor and stator. Rai (1987) simulated incompressible two-dimensional flows in a turbine stage. He solved the thin layer Navier-Stokes equations to obtain the flow around rotor and stator simultaneously and used the sliding mesh method and interpolated the flow properties at the interface. Rai (1987) and Rai and Madaran (1990) simulated three-dimensional flows around rotor-stator blades. Sadri (1997) used control volume approach and PISO method and simulated two-dimensional inviscid and incompressible flow around rotor-stator blades. He used H-type mesh around rotor and stator blades separately. Jorgenson and Chima (1989) considered speudo three-dimensional flow in a stage of a turbo-pump. They used Jameson's method and C-type mesh around each blade. Arnone and Paccini (1996) solved the flow numerically in one stage of PGT2 gas turbine. They also analyzed the interaction of inlet guide vanes and rotor blades for supersonic flow in a compressor Arnone and Paccini (1998). Daneshkhah and Shirani (2001) simulated inviscid compressible flow using Jameson and mesh sliding methods and studied the rotor-stator interactions. Since the flow was assumed ideal and there were no boundary layers, they used uniform grid and the movement of the rotor blade at each time step was a fraction of the mesh size in the span wise direction.

In this paper, the two-dimensional equations of motion were used to consider unsteady, compressible, viscous flow in a stage of a turbomachine by means of control volume approach. A computer code was developed and tested to make sure that it works properly Arnone and Paccini (1998). Because the flow is viscous, non-uniform grid was used to resolve the viscous layer close to the blades. The sliding mesh and interpolation techniques are capable of simulating the flow with unequal pitches of the rotor and stator blades. Four test cases are presented here. The first case is an inviscid flow over a bicircular cascade. The second case is viscous flow in a cascade with NACA0012 blade shapes. The third case is viscous flow in two stationary and parallel cascades with bicircular blades. The last case is viscous flow in a turbine stage with bicircular blades. The result show that the computer program is capable of simulating such flows and the effect of unsteadiness of the flow depends on the rotor linear speed and this effect is much higher on rotor than on the stator blades.

Governing Equations: Since the flow on the turbomachine blades are in such a way that either there are no recirculation zones or very small ones, then the thin layer approximation is used to simplify the equations of motion. The conservative form of the equations for two-dimension, unsteady, compressible, viscous flow in Cartesian coordinate system is:

$$\frac{\partial Q}{\partial t}+\frac{\partial F}{\partial x}+\frac{\partial G}{\partial y}=0 \qquad (1)$$

where:

$$Q=\begin{bmatrix}\rho\\ \rho u\\ \rho v\\ \rho E\end{bmatrix} \qquad F=\begin{bmatrix}\rho u\\ \rho u^2+p\\ \rho uv\\ \rho uH\end{bmatrix} \qquad G=\begin{bmatrix}\rho v\\ \rho uv+\mu\frac{\partial u}{\partial y}\\ \rho v^2+p+\frac{4}{3}\mu\frac{\partial v}{\partial y}\\ \rho vH+\mu u\frac{\partial u}{\partial y}+\frac{4}{3}\mu v\frac{\partial v}{\partial y}+k\frac{\partial T}{\partial y}\end{bmatrix} \qquad (2)$$

ρ is density, u and v are velocity components, respectively, p is pressure, E is total energy, and H is total enthalpy. The fluid is assumed perfect gas and the following state equations were used.

$$p=\rho RT$$

$$E=\frac{p}{(\gamma-1)\rho}+\frac{1}{2}(u^2+v^2) \qquad (3)$$

$$H=E+\frac{p}{\rho}$$

Where, γ is the ratio of specific heat coefficients.

There are four different boundary conditions. They are inlet, outlet, solid and periodic boundary conditions. Since the flow at the inlet is subsonic, three physical properties, the total pressure and temperature, and the flow angle were given at this boundary. The flow at the outlet boundary is also subsonic and the static pressure was given. At the solid boundaries, the blade surface and the relative convective fluxes are zero and the pressure was obtained from the momentum equation in the direction normal to the solid surface. For side surfaces of the flow domain, the periodic boundary conditions were used.

Numerical Method: Since the body fitted grid is used, the equations were first transferred to the body fitted coordinate system. The equations were then integrated over a control volume, grid cell, using Jameson method. In this method, the spatial derivatives are approximated by second order central difference method. To prevent numerical oscillations, the second and fourth order artificial dissipations were used Arnone and Paccini (1998). All the flow properties were defined at the cell centers, collocated grid. The brief Jameson method is presented here.

$$\iint\frac{\partial Q}{\partial t}dxdy+\iint\left(\frac{\partial F}{\partial x}+\frac{\partial G}{\partial y}\right)dxdy=0 \qquad (4)$$

where,

$$\iint\left(\frac{\partial F}{\partial x}+\frac{\partial G}{\partial y}\right)dxdy=\oint(Fdy-Gdx) \qquad (5)$$

and

$$\iint\frac{\partial Q}{\partial t}dxdy=\frac{\partial Q}{\partial t}S \qquad (6)$$

Where, S is the area of control volume.

The above equations were then integrated with respect to time and the fourth order Runge-Kutta method was used. Since the unsteady solution is required, the time step, Δt_c, was chosen to be the minimum of allowable local time steps, Δt_{local}. That is,

$$\Delta t_c=\min(\Delta t_{local}) \qquad (8)$$

Grid Sliding: To analyze the flow in one stage of a turbomachine, which includes rotor and stator rows, several different methods are used to obtain the solution at the interface of the moving and stationary blades, properly. Among the methods are used for this purpose, the following methods described here are the most important ones. In the mixing plane method, the flow variables are averaged at the interface of rotor and stator blades. In the average path method, each blade row is considered separately and their interactions are modeled. In sliding mesh method, the grid around each blade row is separately generated and at the interface, there is relative motion of the grids. This method is more accurate and was used in this paper. The sliding mesh method itself is classified by the following four methods. 1) Overlaid method, where there are common region of the two sets of moving and stationary grids, 2) Sliding interface method, where there are a common surface between the two meshes, 3) Shearing interface method, where there are a zone between the two grid sets and the cells created by connecting the nodes of the two grid sets at the zone between them deform as one set of the grid moves, 4) Distorting treatment method, there are one grid for the whole flow with three regions, stator, rotor, and space between them where the cells are distorted and moved to their previous positions periodically.

In this paper, the sliding mesh with the sliding interface is used. At each time step, the grid around the rotor moves by amount of Δs,

$$\Delta s = v_b . \Delta t_c \tag{7}$$

Where, v_b is the rotor speed, and Δt_c is the time step. Since the rotor moves, the grid cells at the interface do not coincides with each other (Fig.1), and we need to use some approximations to obtain the fluxes at the interface. The interpolation method is used at the sliding surface, is such that it is conservative and third order accurate.

The grid generated around each row is H-type and obtained algebraically.

Results and Discussion

To examine the ability of the method and integrity of the computer code and to analyze the flow in one stage of a turbomachine, the following four cases were simulated.

a. Steady Ideal Flow in a Bicircular Cascade: Ideal flow in a bicircular cascade was simulated for two different back static pressures. The isentropic exit Mach numbers, M_{2is}, for these cases are 0.675 and 0.73. Fig. 2 shows the geometry and the computational grid. The number of grid points is128×52. Fig. 3 shows the velocity residual for three different mesh sizes, when M_{2is}= 0.73. The grid study shows that the grid size 128×52 is fine enough to obtain acceptable results. Fig. 4 shows the distribution of Mach number at the wall and at the line of symmetry for two cases, M_{2is}= 0.675 and M_{2is}= 0.73. The results were compared with other numerical results, ref.'s (Shirani and Zirak, 2001) and (Arnone and Swanson, 1993), and good agreement were obtained. Fig. 5 and 6 show the pressure and Mach contours, respectively. These figures show the location and properties of the shock wave. The results are fine and there are no numerical oscillations. Comparing the results for two Mach numbers, it is shown that by reducing the back static pressure (or increasing the exit Mach number), the shock wave becomes stronger and it moves toward the trailing edge of the blade.

b. Steady Viscous Flow in a NACA0012 Cascade: Fig. 7 shows the geometry and the computational grid. After the grid study, it was found out that the proper grid size is158×61. The inlet stagnation pressure and outlet static pressure are chosen such that the exit isentropic Mach number is M_{2is}= 0.73. Fig. 8 show the inlet velocity residual. As can be seen, the solution is converged. Fig. 9 and 10 show the static pressure at the surface and the Mach number distribution. In this figure, the location of shock wave, changes of the pressure and Mach number across the shock wave are shown. Fig. 11 and 12 show the contours of pressure and Mach number. The solution contains no numerical oscillations and is according to the physics of the flow.

c. Steady Viscous Flow on the Two Stationary Bicircular Blades: In order to examine the ability of computer code for analysis of the flow in two sets of grids, the flow in two stationary cascades with bicircular blades are examined. Fig. 13 shows the geometry and the computational grid. After the grid study, it was found out that the proper grid size is 317×61. The inlet stagnation pressure and outlet static pressure are selected such that the exit isentropic Mach number is M_{2is}= 0.391. Fig. 14 and 15 show the velocity residual at two domains. Each domain contains one cascade. Fig. 16 and 17 show the pressure and Mach number contours, respectively. The results at the interface are smooth and fine.

d. Unsteady Viscous Flow in a Bicircular Stage: A turbomachine stage containing two rows of stator and rotor with bicircular blades is considered here. The rotor along with its surrounding computational grid moves with constant speed. The interface between the rotor and stator is a sliding surface, which is treated by the sliding interface method. The geometry and grid configuration are the same as the previous case and are shown in Fig. 13. Since the flow is unsteady, the convergence criterion in this problem is different from previous cases. In fact, the flow in this problem is periodic and it should be repeated after some interval of time. So the numerical results are converged when solution becomes periodic. Thus after some initial iterations, the numerical solution should be converged and all the flow variables must be repeated after each period. Since the lift force is more sensitive to the solution, we chose this parameter to check the convergence. Fig. 18 and 19 show the lift force applied on stator and rotor blade during the first four period of the numerical solution. As shown the periodic solution is obtained after initial oscillations. This figure is for rotor velocity being 10 m/sec. For this value of velocity, the computational time for each period is very large and is about 40 hrs. Fig. 20 and 21 show the lift force applied on stator and rotor blade for the case when the rotor velocity is 25 m/sec. In this case the computational time reduces to about 20 hrs for each period. In these figures, the solution for the first six periods is shown. It is clear that the solution has converged. In this case, the effect of rotor-stator interaction is stronger and this effect is more on the rotor blades than on the stator blades. Fig.' 22 and 23 show the mean, maximum and minimum of pressure distribution on the stator and rotor blades during one period, respectively. These results are for v_b = 25 m/sec. Because the blades are symmetric, the pressure distributions on both sides of the stator blades are almost the same. At the trailing edge of the stator, the change in the pressure is stronger, because this part of the flow field is more affected by the interface. But the changes of pressure on the rotor are almost the same in all parts of rotor and it is stronger than in the stator. Fig. 24 and 25 show the pressure contours at five different times during one period for v_b = 10 and 25 m/sec, respectively. The location of maximum and minimum pressures on the blade surfaces are the same for both cases, but the magnitude of the changes of pressure are higher for v_b = 25 m/sec.

Fig. 26 and 27 show the contours of Mach number for five different times during one period for v_b = 10 and 25 m/sec, respectively. These figures show the effect of unsteadiness of the problem on the flow especially at the interface region. It also can be seen that the continuity of the solution at the sliding mesh surface is very well and there is no numerical oscillations. The comparison of the two solutions show that, as rotor velocity increases, the unsteady effect are stronger and the wake region of the rotor blades are grown faster, which can affect the next stator blades in the turbomachines.

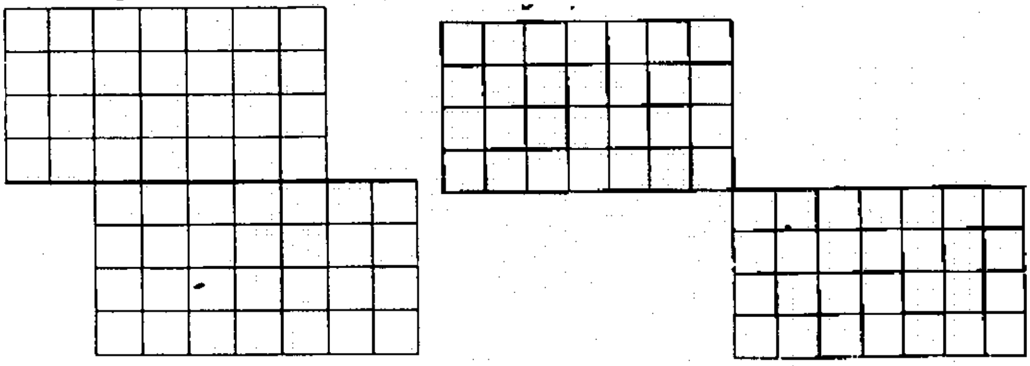
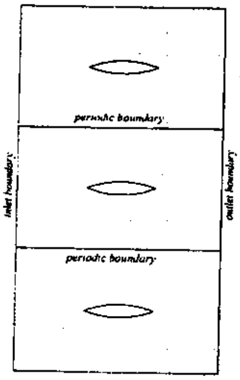
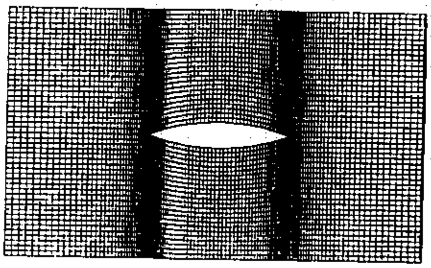


Fig. 1: Computational Grid Sliding



(الف)



(ب)

Fig. 2: Bicircular Cascade and Grid

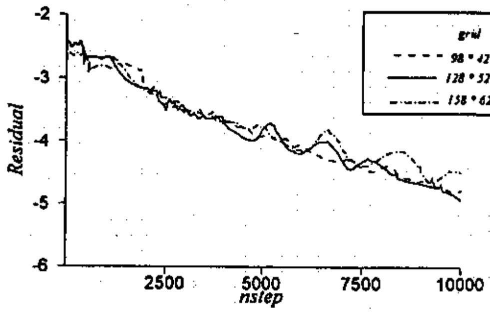
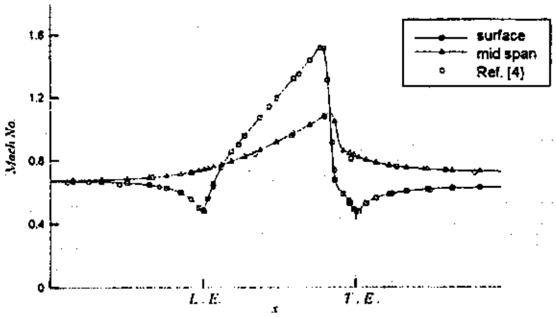
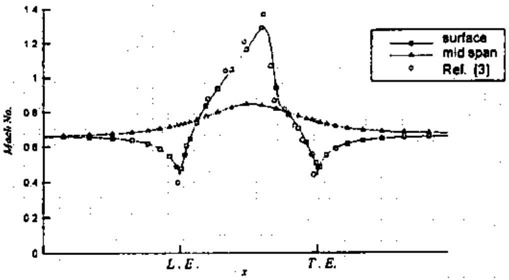


Fig. 3: Velocity Residual for Three Grids, M_{2is}=0.73

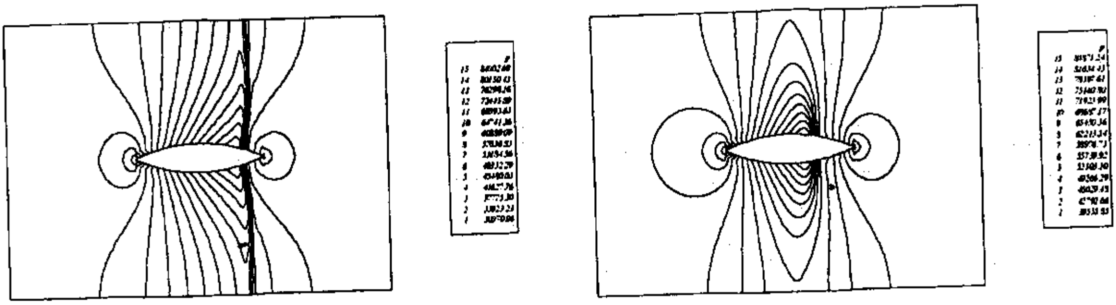


M_{2is}=0.73 (b



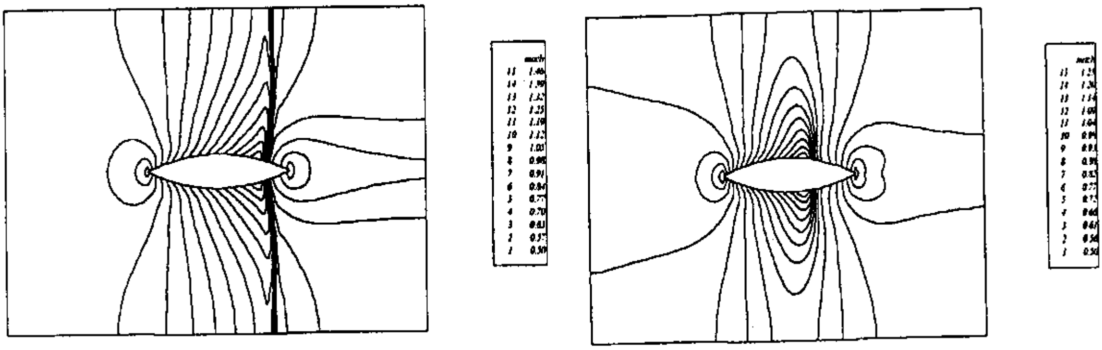
M_{2is}=0.675 (a

Fig. 4: Mach Number Distribution at Wall and Axis of Symmetric



M_{2is}= 0.675 (b M_{2is}= 0.675 (a

Fig. 5: Contours of Pressure



M_{2is}= 0.675(b M_{2is}= 0.675 (a

Fig. 6: Contours of Mach Number

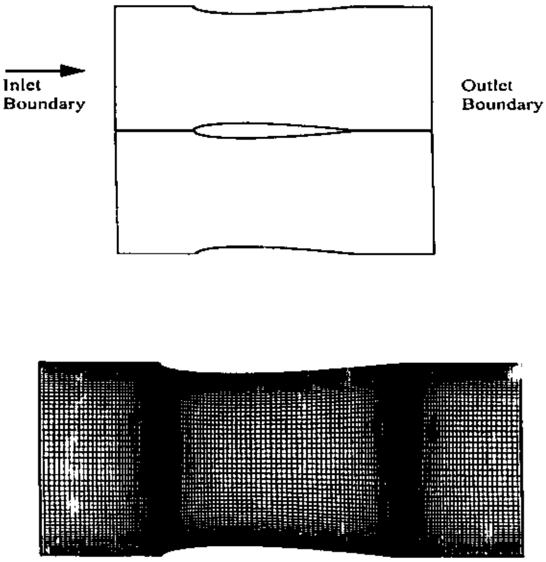


Fig. 7: NACA 0012 and its Computational Grid

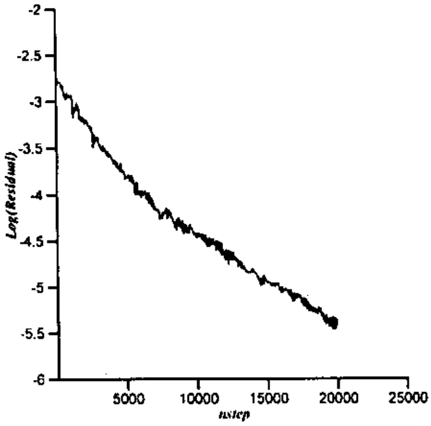


Fig. 8: Entrance Velocity Residual

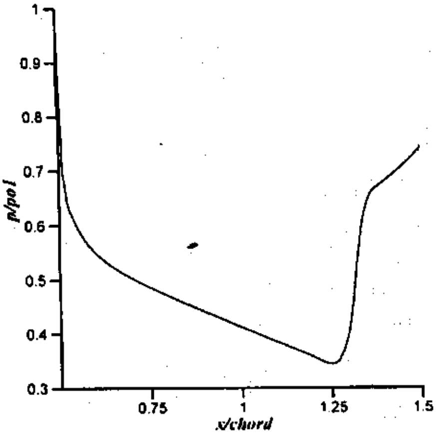


Fig. 9: Pressure Distribution on Airfoil

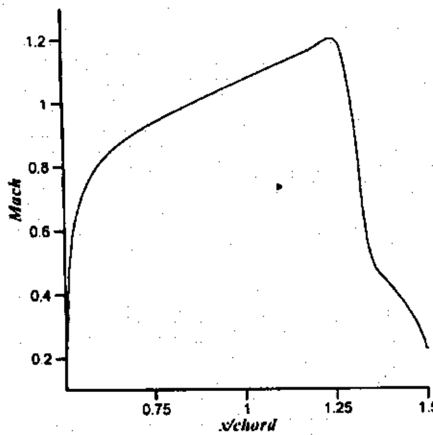


Fig. 10: Mach Number Distribution on Airfoil

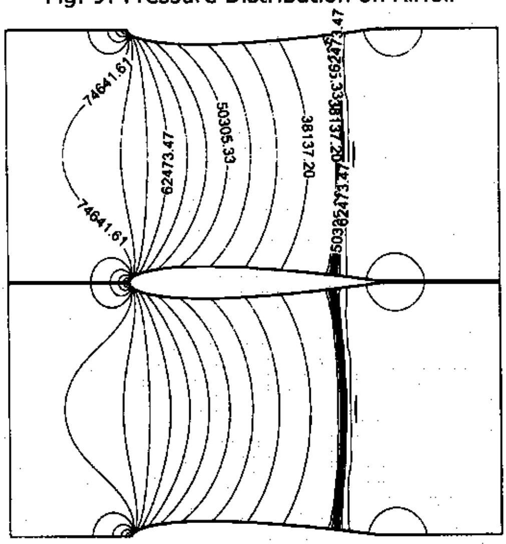


Fig. 11: Pressure Contours

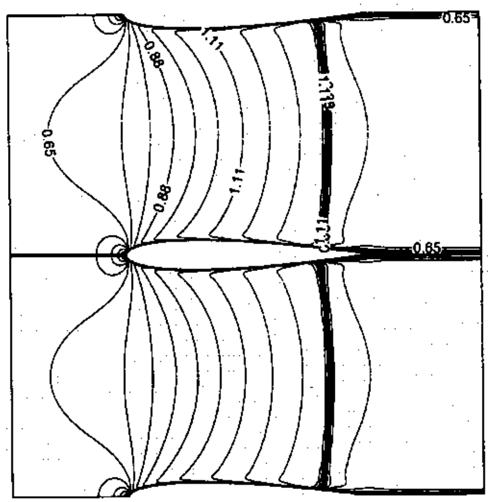


Fig. 12: Mach Number Contours

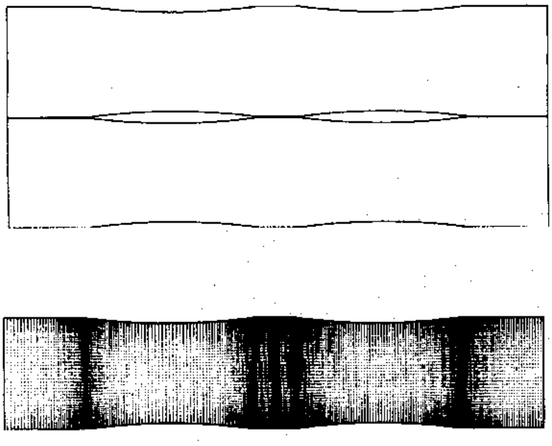


Fig. 13: Geometry and Computational Grid

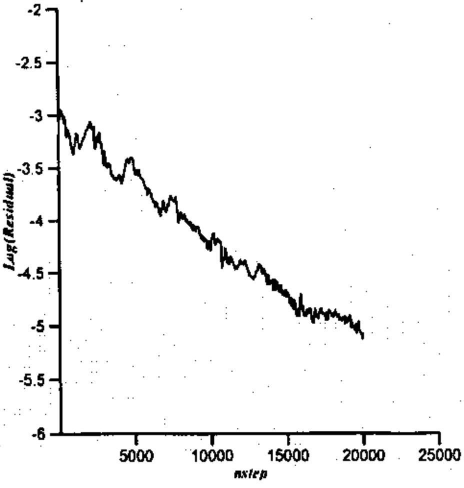


Fig. 14: Velocity Residual for the First Row

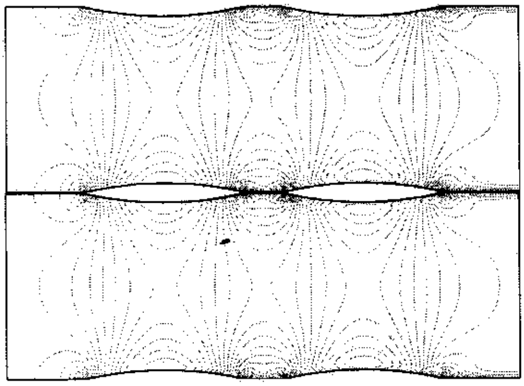


Fig. 15: Velocity Residual for the Second Row

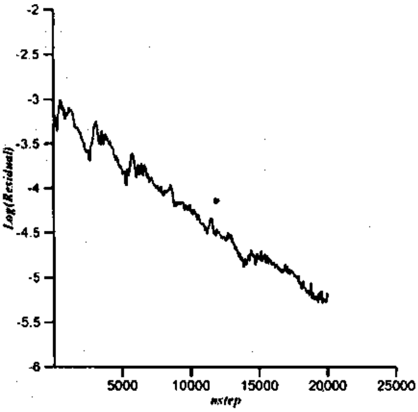


Fig. 16: Pressure Contours

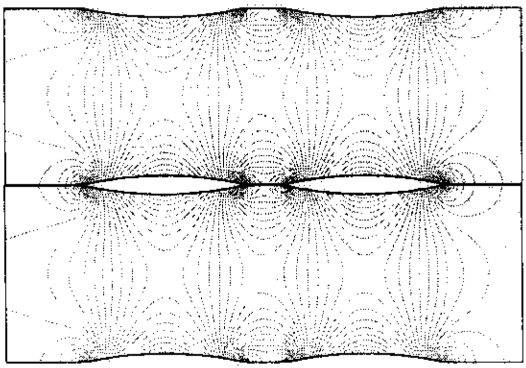


Fig. 17: Mach Number Contour, v_b = 10m/s

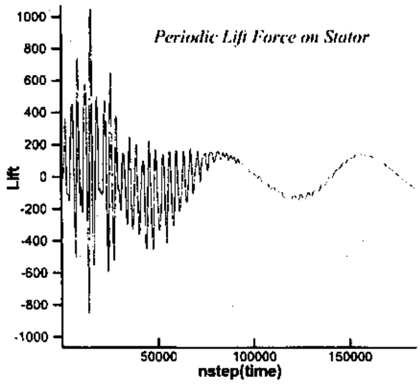


Fig. 18: Lift Force on Stator, v_b = 10 m/s

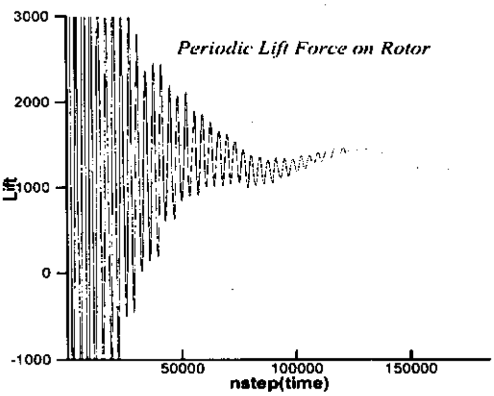


Fig. 19: Lift Force on Rotor, v_b = 10 m/sec

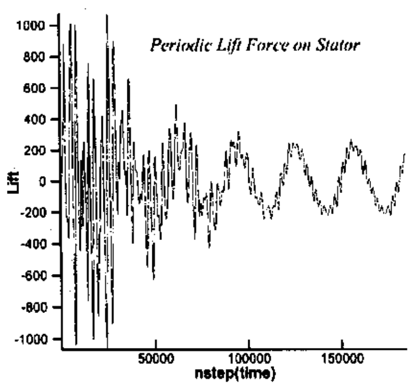


Fig. 20: Lift Force on Stator, v_b = 25 m/sec

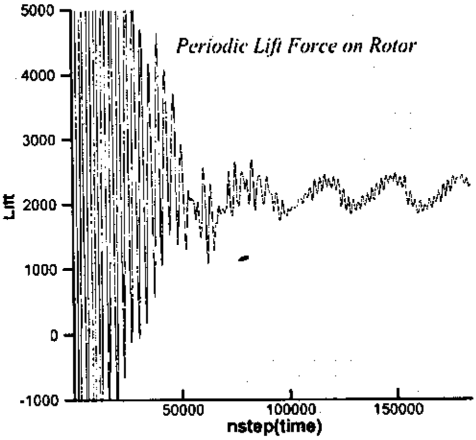


Fig. 21: Lift Force on Rotor, v_b =25m/s

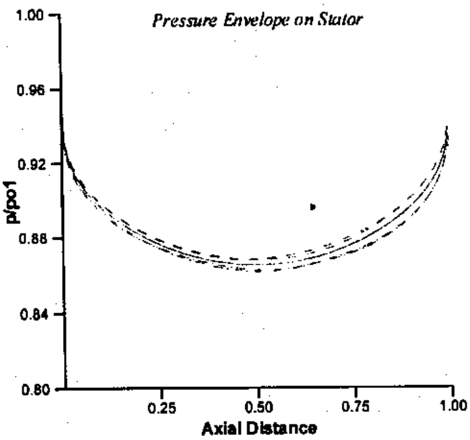


Fig. 22: Pressure Distrib. on Stator, v_b=25 m/s

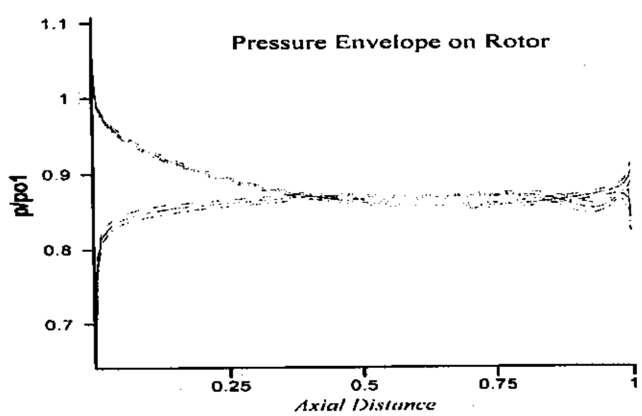


Fig. 23: Pressure Distribution on Rotor, v_b = 25 m/sec

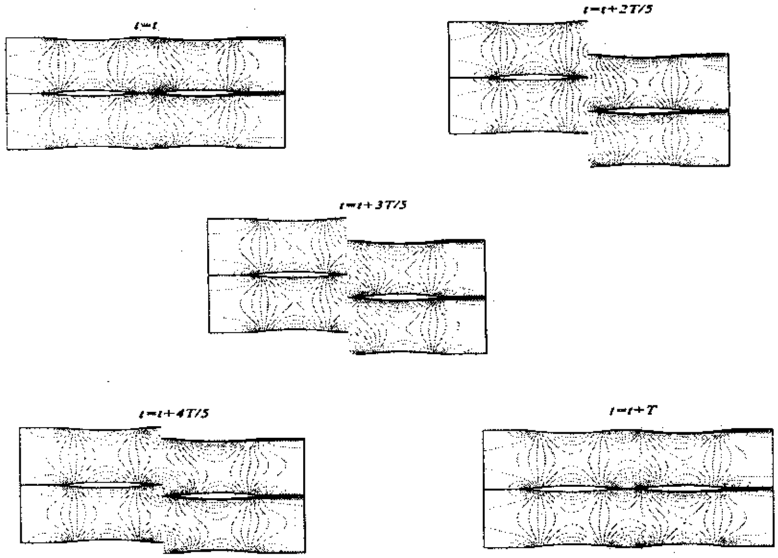


Fig. 24: Pressure Contours, v_b = 10 m/sec

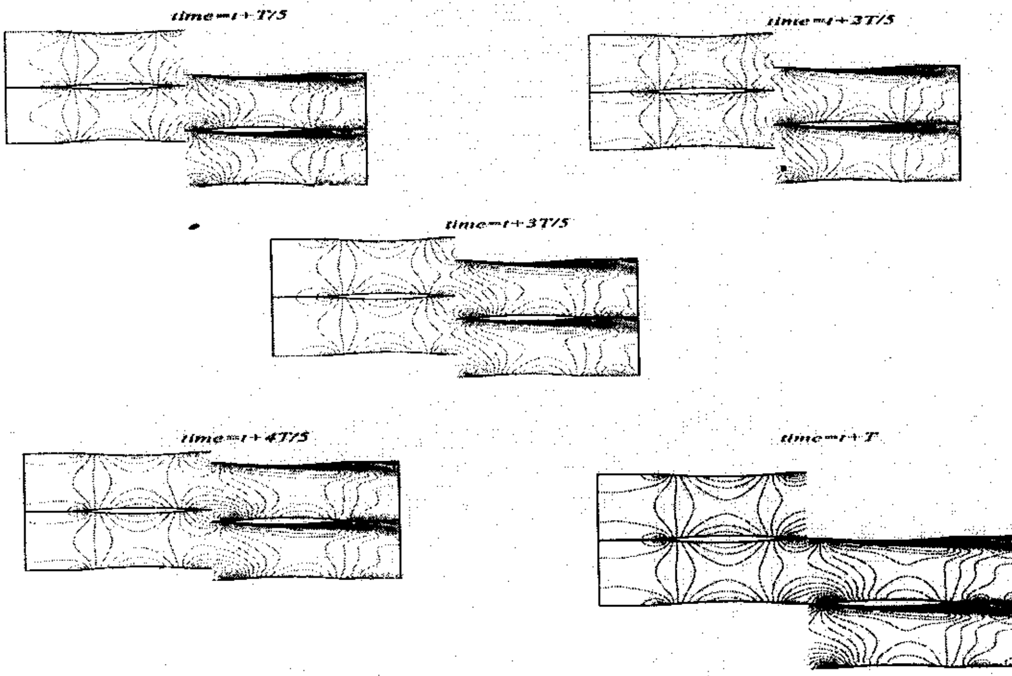


Fig. 25: Pressure Contours, v_b = 25 m/sec

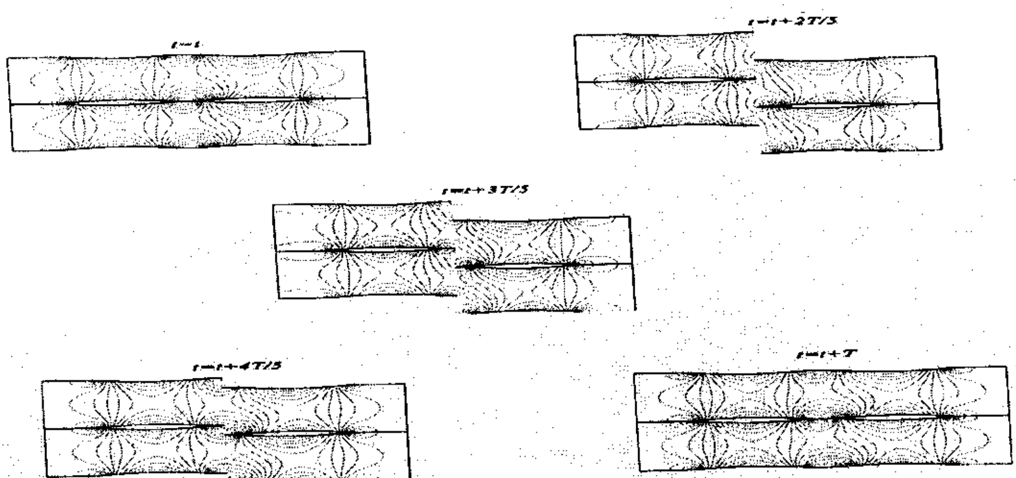


Fig. 26: Mach Number Contours, v_b = 10 m/sec

Conclusion

The integration of control volume method of Jameson was used in this work to simulate the viscous, unsteady, compressible, two-dimensional flow in one stage of a turbomachine. At the interface, sliding mesh method and a conservative third order interpolation technique, the cubic Spline method, was used to calculate the fluxes. This kind of interpolation was first used in this paper for simulating the viscous flow at the interface and has shown that it gives very smooth results. The results show no oscillations at the interface. The computer code developed for this flow was tested for four different cases and it shows that the results are in good agreement with others results. So the method used in this paper and the computer ode provided are capable of simulating such flows.

The rotor-stator interaction, affects the lift force, pressure distribution, and velocity field especially on

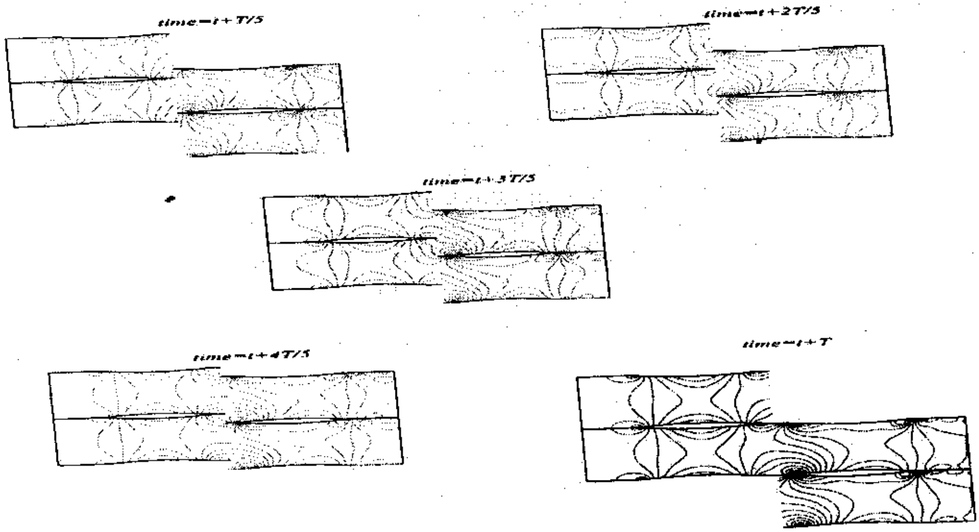


Fig. 27: Mach Number Contours, v_b = 25 m/sec

the rotor blades. These effects become stronger when the rotor speed increases.

Acknowledgement: This work was supported by and carried out at Isfahan University of Technology (IUT) and supported by research funds of the university. The authors would like to thank IUT for its support.

References

Arnone, A. and R. C. Swanson; 1993. "A Navier-Stokes Solver for Turbomachinery Applications," ASME J., V. 115.

Arnone, A., and R. Paccini; 1996."Rotor-Stator Interaction Analysis Using the Naveir-Stokes Equations and a Multi-Grid Method," ASME J. of Turbomachinery, V. 118.

Arnone, A., and R. Paccini; 1998."IGV-Rotor Interaction Analysis in a Transonic Compressor Using the Naveir-Stokes Equations," ASME J. of Turbomachinery, V. 120.

Barnett, M., D. E. Kobbs and D. E. Edwards; 1990. "Inviscid-Viscous Interaction Analysis of Compressor Cascade Performance," ASME GT-90-15.

Crawford, M.E. and W. M. Kays; 1976. "A Program for Numerical Computation of Two-Dimension," NACA CR2742.

Denton, G.D., 1975. "A Time Marching Method for Two and Three- Dimensional Blade to Blade Flows," Aeronautical Research Council, R&M, 3775.

Daneshkhah, K., and E. Shirani; 2001. "Rotor-Stator Interaction," Journal of Daneshkade Mohandesi, Vol. 13, No. 47.

Dering, R.P., H. D. Joslyn, L. W. Hardin and J. H. Wagner; 1982. "Turbine Rotor-Stator Interaction," ASME J. of Eng., for Power, V. 104.

Erdos, J.I., E. Alzner and W. McNally; 1972. "Numericval Solution of Periodic Transonic Flow Through a Fan Stage," AIAA J., V. 15, No. 11.

Jameson, A., A. Rizzi , W. Schmidt, and D. Whitefield; 1981. "Finite Vol. Solution for the Euler Equation for Transonic Flow over Airfoil and Wing Including Viscous Effects," AIAA Paper, 81-1265.

Jameson, A., W. Schmidt and E. Turkel; 1981. "Numerical Solution of Euler Equations by Finite Vol. Method Using Runge-Kutta Time-Stepping Scheme," AIAA Paper No. 81-1259.

Jameson, A., 1991. "Time Dependent Calculation Using Multigrid with Application to Unsteady Flow Past Airfoil and Wings," AIAA Paper No. 91-1596.

Jorgenson, C.E. and V. Chima; 1989. "Explicit Runge-Kutta Method for Unsteady Rotor-Stator Interaction," AIAA J., V. 27, No. 6.

Knight, C.J., and D. Choi 1989. "Dev. of a Viscous Cascade Code Based on Scalar Implicit Factorization," AIAA J., V. 27, NO. 5.

Lakshminarayana, B., and T. R. Govindan; 1981. "Analysis of Turbulent Boundary Layer on Cascade," AIAA J. V. 19.

Li, J.S., P. N. Sockol and J. M. Prahl; 1989."Navier-Stokes Cascade with a Stiff k-e Turbulence Solver," AIAA Paper, 88-0594.

McDonald, P.W., 1971. "The Computation of Transonic Flow Through two-Dimensional Gas Turbine Cascade," ASME paper 71-gt-89.

Rai, M.M., 1987. "Navier-Stokes Simulation of Rotor-Stator Interaction Using Pateheld and Overlaid Grid," AIAA J. of Propulsion and Power, V. 3, No. 5.

Rai, M.M., 1987. "Unsteady 3-D Navier-Stokes Simulation of Turbine Rotor-Stator Interaction," AIAA Paper, No. 87-2058.

Rai, M.M. and N. K. Madaran; 1990. "Multi-Airfoil Navier-Stokes Simulation of Turbine Rotor-Stator Interaction," ASMER J. of Turbomachinery, V. 112.

Sadri, M.A., 1997. "Numerical Modeling of Rotor-Stator Interaction in Turbomachinery," PhD Thesis, Univ. of London.

Stager, J.L., T. H. Pullium and R. V. Chima; 1980. "An Implicit Finite Difference Code for Inviscid and Viscous Cascade Flow," AIAA Paper, 80-1427.

Shamroth, S. J., H. McDonald and W. R. Briley; 1982. "Application of Navier-Stokes Analysis of Transonic Cascade Flow Fields," ASME J. of Eng. for Power.

Shirani, E. and S. Zirak; 2001. "Numerical analysis of Compressible Steady/Unsteady, and Inviscid/Viscous Flow in Cascades and Effects of Viscosity on the Flows," Amirkabir J., Vol. 12, No. 47.