9th ANKARA INTERNATIONAL AEROSPACE CONFERENCE **AIAC-2017-105**
20-22 September 2017 - METU, Ankara TURKEY

DEVELOPMENT OF A BLADE TO BLADE SOLVER FOR AXIAL TURBOMACHINERY

Mustafa BİLGİÇ[1]
Middle East Technical University
Ankara, Turkey

M Halûk AKSEL[2]
Middle East Technical University
Ankara, Turkey

ABSTRACT

The blade to blade streamsurface, one of the stream surfaces defined by [Wu, 1952], models the flow in m-rθ plane assuming that flow is axisymmetric. Two different approaches are used for the solution of the two dimensional Euler equations through blade to blade streamsurface. The first one includes the solution of the steady form of the Euler equations. Artificial compressibility concept is added to cover transonic flow. The resultant system of equations is solved using modified tri diagonal matrix algorithm. The second method includes the solution of the unsteady form of the Euler equations. The face fluxes are evaluated using upwind approach. At the inlet and outlet, characteristic boundary conditions are applied. On the solid wall, slip conditions are imposed. Periodic boundary conditions are applied upstream and downstream of the blade passage. Flow variables are integrated in time explicitly.

INTRODUCTION

Compressor and turbine are the two major components of a typical gas turbine because work addition and extraction occurs within these components. Since the compressor adjusts the pressure ratio and mass flow, it has great influence on the characteristics of a gas turbine. The turbine extracts work from high pressure and high temperature air flow. The design process of the compressor and turbine starts with the cycle analysis. The output of the cycle analysis is used in the mean-line design in order to determine the initial sizing of the gas path and number of stages. The output of the mean-line analysis feeds the detailed airfoil design procedure. The design of blade is highly complicated process because of the presence of extremely complex flows. These complex flow structures include three dimensional boundary layers, mixture of different types of flows, hub and tip boundary layers, wakes, secondary flow vortices and unsteadiness. Therefore, fast but accurate blade design tools which could handle all of these flow types are needed. In the gas turbine industry, these design methodologies highly depend on two or/and three dimensional CFD analysis.

The application of the first numerical methods in the turbomachinery was seen in 1940s. The very first solution of turbomachinery flow was the solution of the three dimensional potential equation [Wu, 1952]. However, the potential equation does not include rotationality, non-uniformity in enthalpy and entropy generation across the shock waves. Some of these difficulties were eliminated with introduction of S_1 and S_2 streamsurfaces by [Wu, 1952]. The problem was formulated using the definition of the streamfunction on the streamsurfaces for both absolute and relative frames of reference. The equation of continuity and momentum for

[1] Phd Student, , Email: mustafa.bilgic_02@metu.edu.tr
[2] Prof. in a Department of Mechanical Engineering, Email: aksel@metu.edu.tr

fluid flow are combined into one principal equation. The characteristic of the principal equation depends on the local Mach number. Definition of two stream surfaces, which are blade to blade (S_1) and through flow (S_2) streamsurfaces, was regarded as the complete three dimensional solution of the problem (Q3D-Quasi Three Dimensional Flow). In this model, there are some assumptions to simplify the problem further. As the first and main assumption, flow field has periodic unsteadiness leading to steady flow analysis in both absolute and relative frames of reference. Also, the flow through the S_2 plane is assumed to be axisymmetric and S_1 streamsurface is assumed as the surface of revolution. Since some assumptions are included, it is necessary to check the accuracy of the method with the experimental results. The Q3D method is quite fast as compared to the complete three dimensional analysis.

Newton Solver

In this study, two different approaches are used for the solution of the blade to blade flow. The first one is the steady solution of two dimensional Euler equations [Giles, 1985]. Integral form of the two dimensional Euler equations in a rotating frame of reference are solved by using cell centered finite volume method. The flow domain is discretized on the intrinsic streamline grid. Initially Euler equation is discretized as x and y momentum equations. After some manipulation, the streamwise and normal momentum equations are obtained from momentum equations in x and y directions. For transonic flow solution, the artificial compressibility is implemented and shocks are captured accurately. After the discretization is completed, the Newton-Raphson method is applied to the discrete flow equations. Together with the linearization, some manipulation is applied using continuity and energy equations in order to reduce the number of unknowns to two. After the linearization is applied to all discrete control volumes, a huge set of linear system of equations is formed. For the solution of linear system of equations, a direct approach, which is the solution of the resultant linear system of equations using modified tri-diagonal algorithm [Thomas, 1949], is used. The density and nodal coordinates are updated for the next iteration level using under relaxation. The relaxation factor is used in order to prevent excessive changes in density and nodal coordinates.

Time Marching Solver

The second approach for the solution of the Euler equation is the time marching finite volume method. The integral form of the unsteady Euler equations are discretized on a polyhedral control volume. The flow variables are stored at the centers of the control volumes (cell centered scheme). Fluxes are evaluated at the faces of the each cell considering the left and right states of the faces. The left and right states of the faces are determined using MUSCL approach with modified Van Albada Limiter function [Blazek, 2001]. In order to calculate face flux values, upwind discretization is used. The time integration is carried out explicitly by using the Runge-Kutta time integration [Hirsh, 1988]. In order to accelerate the convergence, local time stepping is applied.

COMPUTATIONAL DOMAIN AND SOLUTION METHODS

In this section, details of the blade to blade computational domain is described. Generation of the computational domain, discretization, description of the source terms and applied boundary conditions are presented.

Generation of Computational Space

The domain, which is solved as a blade to blade plane, is shown in Figure 1 [Drela, 2008]. The blade to blade surface is assumed to be a surface of revolution. The blade profile is obtained by cutting the blade along the streamline. The resulted two dimensional computation domain is known blade to blade surface which is also known as m – rθ plane. Definition of the streamline comes from the throughflow solver. Throughflow solver generates streamlines as a result of the solution of axisymmetric momentum equations in r – z plane. In addition to the definition of the streamline, the streamtube extraction/contraction also comes from throughflow solver.

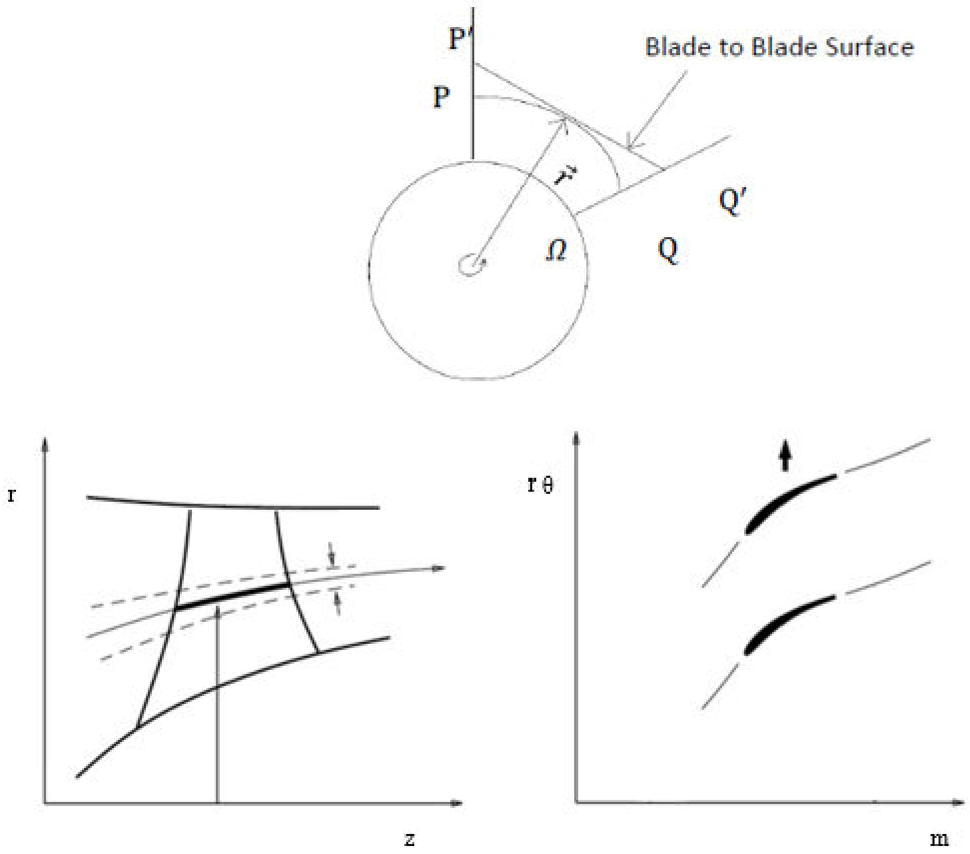


Figure 1 Computational Domain [Drela, 2008]

Discretization

After the solution domain is generated, the spatial discretization is made on structured grid. In order to generate grid, two dimensional elliptic grid generation method is used. Two different methods are used in order to solve the discretized Euler equations. One method solves the steady form of the Euler equations together with the artificial compressibility. Energy equation is simply conservation of the rothalpy.

$$\oint \rho\vec{V}.\hat{n}ds = 0 \tag{1}$$

$$\oint \left(\rho\left(\vec{V}.\hat{n}\right)\vec{V} + P\hat{n}\right)ds = \int \rho\vec{f_b}d\forall \tag{2}$$

$$\oint \rho\vec{V}.\hat{n}Ids = 0 \tag{3}$$

where, ρ is the density, V is the relative velocity, n is the unit vector normal to the cell surface, P is the pressure, ds is the surface length of the cell, ∀ is the volume of the cell, f_b is body force and I is the rothalpy. The relationship between total enthalpy and rothalpy is given as

$$I = h_t - \frac{(\Omega r)^2}{2} \tag{4}$$

The details of the linearization and manipulation of the equations are not presented here since it is a very long process. The details of this process can be found in reference [Bilgic, 2015]. The second method solves the unsteady Euler equations on *m*-rθ plane. The vector form of this equation can be written as

$$\frac{\partial}{\partial t}\int \vec{W} d\forall + \oint \vec{F_c} dS = \int \vec{Q} d\forall \tag{5}$$

where, t is time, W is conservative variable vector, Fc is convective flux vectorand Q is the source term vector. The details of the vector components can be found in any Computational Fluid Dyanmics (CFD) books. Since the equations are hyperbolic in time, the flow variables are marched in time. For the time integration, 4 stage Runge-Kutta time integration method is used.

Source Terms

The solution of the rotor requires the effects of the Coriolis and centrifugal acceleration. These effects are introduced by the source term. The contents of Q in equation (5) and f_b in equation (2) are the same. The rotational effects are projected on *m*-rθ plane using some geometrical manipulations. The resulting body forces in m and θ direction can be written as

$$f_{b_m} = -2\Omega V\, sin\beta\, sin\phi + \Omega^2 r\, sin\phi \tag{6}$$

$$f_{b_\theta} = 2\Omega V\, cos\beta\, sin\phi \tag{7}$$

where, Ω is the rotational speed, β is the flow angle in relative frame, r is the radius measured from center of rotation and Φ is the streamline curvature angle.

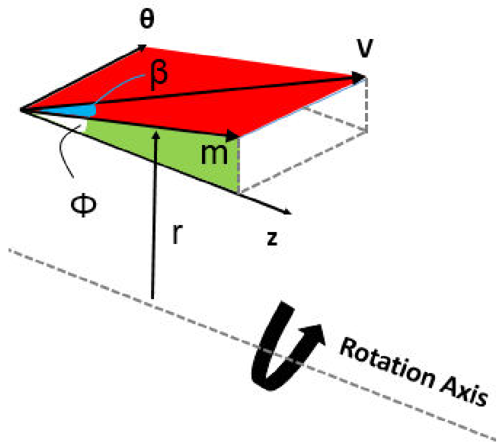


Figure 2 Terms appearing in B2B domain

Boundary Conditions

The Newton solver imposes two different boundary conditions for unchoked and choked flows. If the flow is subsonic, simply Mach number is specified at the inlet plane. Once the Mach number is given, the mass flow rate is determined at the inlet. When the mass flow is fixed, it is not necessary to specify a variable at the exit plane for subsonic flow. However, when the flow is choked, the back pressure must be specified at the exit plane. Furthermore, if flow is supersonic at the inlet, specifying Mach number is not sufficient. Implementation of supersonic inflow boundary condition is not included in this study. On the other hand, time marching solver imposes characteristic boundary conditions at the inlet and exit plane. Therefore, choking of the flow does not necessitates changing the specified variables at the inlet or exit boundaries. The remaining boundaries are the same for both solvers. On the blade surface solid wall boundary condition is imposed while periodic boundary conditions are imposed before and after the blade.

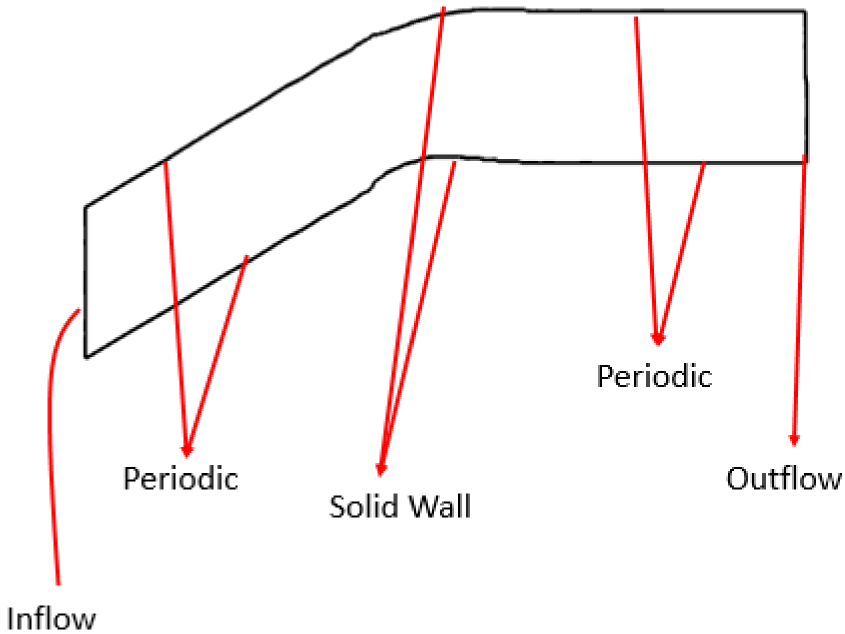


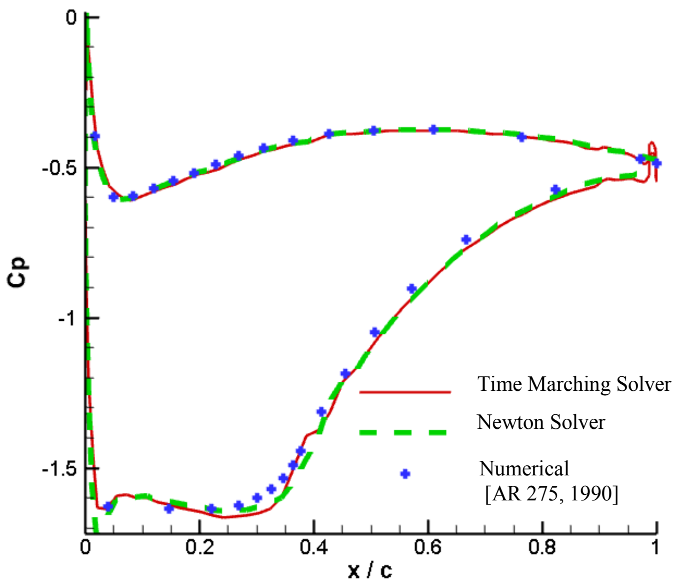
Figure 3 Boundary conditions

RESULTS

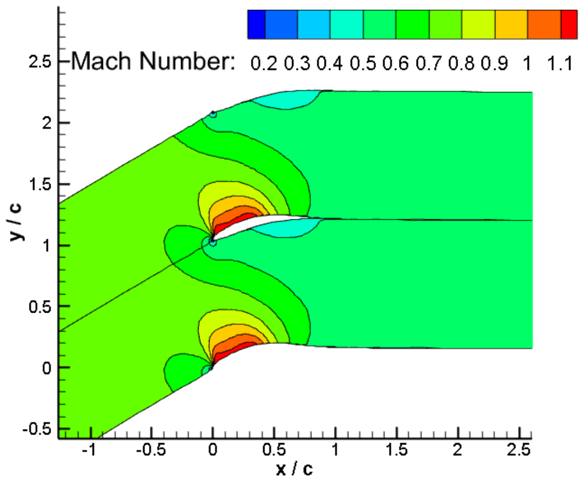
The solvers are tested with various test cases. The results of the solvers are compared with two analytical results and one experimental result.

Sanz Supercritical Compressor Cascade – Analytical Case

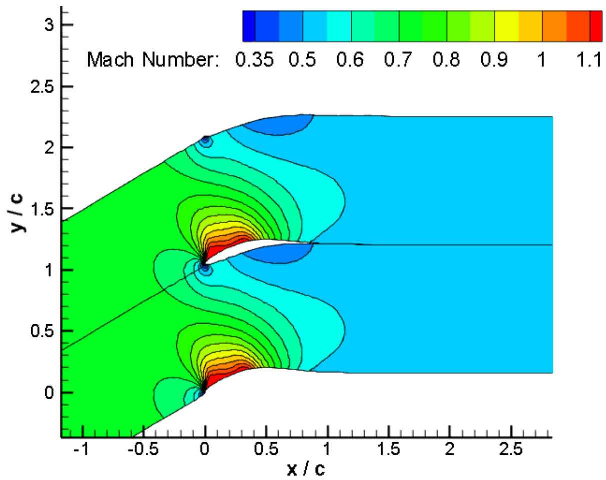
The first case is the supercritical compressor cascade case which is taken from AGARD [AR 275, 1990]. The inlet Mach number and flow angle are 0.711 and 30.81 degree respectively. The exit Mach number and flow angle are 0.544 and -0.35 degrees respectively. The reference result is obtained from the hodograph method. The results of the solvers are compared presenting both pressure coefficient and Mach number contours in Figure 4. The results agree well with the hodograph solution. The pressure ratios obtained from the time marching, Newton and hodograph method are 1.143, 1.144 and 1.145, respectively. At the trailing edge, Newton method imposes Kutta condition. Therefore, the pressure values on suction and pressure sides are compatible. However, a fluctuation seems in the result of time marching solver. Since the Kutta condition cannot be applied in the marching method, this fluctuation appears to be inevitable [Denton, 1999].



a) Comparison of Cp values obtained from solvers with the result of hodograph method



b) Mach number contours obtained from time marching

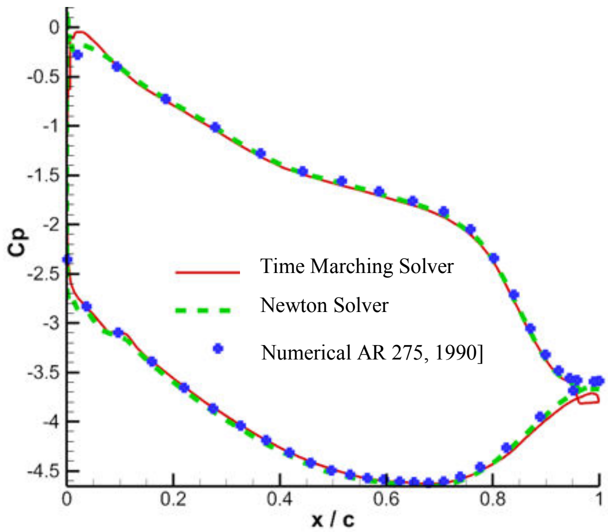


c) Mach number contours obtained from the Newton solver

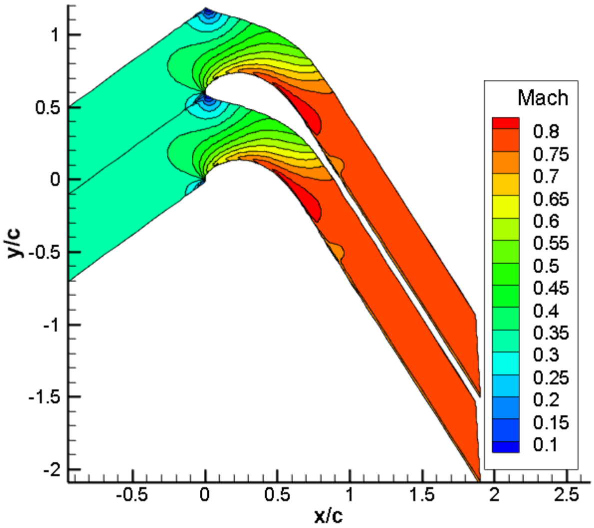
Figure 4 Comparison of the result of the solvers with reference results and Mach number contours – supercritical compressor cascade

Sanz Subcritical Turbine Cascade – Analytical Case

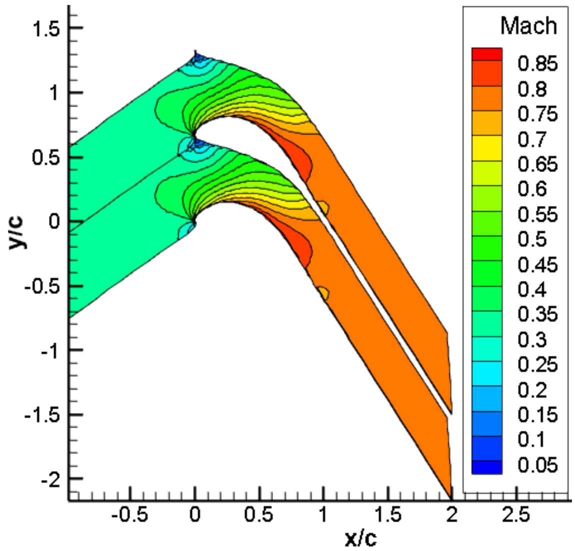
The second case is the subcritical turbine cascade case which is taken from again [AR 275, 1990]. The inlet Mach number and flow angle are 0.343 and 36.0 degree respectively. The exit Mach number and flow angle are 0.765 and -57.35 degrees respectively. The reference result is again obtained from the hodograph method. The results of the solvers are compared using pressure coefficients in Figure 5. The results agree well with the hodograph solution. Trailing edge Kutta condition problem for the time marching solver again appears. The outlet flow angles obtained from the time marching and Newton methods are -57.42 and -57.38 degrees, respectively.



a) Comparison of Cp values obtained from solvers with the result of hodograph method



b) Mach number contours obtained from time marching solver

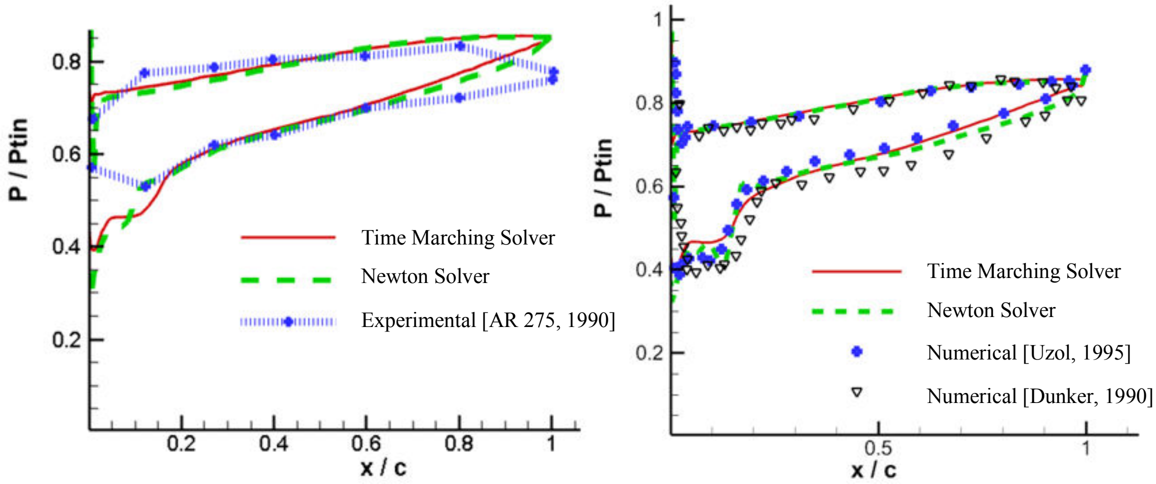


c) Mach number contours obtained from the Newton solver

Figure 5 Comparison of the result of the solvers with reference results and Mach number contours – subcritical turbine cascade

R030 Compressor Rotor – Experimental Case

The third case is one stage compressor experiment taken from [AR 275, 1990]. In order to test the solvers for rotor case, the 18% height of the blade section is selected as the computation domain. Inlet Mach number of the rotor blade is 0.9215. Inlet flow angle and streamline curvature angle are 54.60 and 11.83 degrees, respectively. The pressure ratio of the compressor blade is 1.43. The pressure ratios obtained from the time marching method, Newton method and experiments are 1.453, 1.445 and 1.43, respectively. The pressure distribution of the blade is obtained from experimental results. The results have shown that both Newton and time marching solvers give quite accurate results. In Figure 6a, pressure distribution obtained from solvers agree with the results of experiment published in [AR 275, 1990]. Also the results are compared with the some other numerical results as shown in Figure 2b. The sudden acceleration at the leading edge of the rotor is the common behavior of the numerical results [Dunker, 1990], [Uzol, 1995]. Also the effect of the viscosity in the experimental results at the trailing edge can be seen clearly. However, the Euler solvers retain the pressure rise at the trailing edge.



a) Comparison with experimental results b) Comparison with numerical results

Figure 6 Comparison of the result of the solvers with reference results

CONCLUSION

In this study, in order to solve blade to blade flow through the axial turbomachinery, two different approaches are used. The results has shown that both methods have no superiority over each other in terms of accuracy. The Newton solver has some stability problems for blunt leading edges. Since the Newton solver recalculate the location of the grid points at each iteration (intrinsic streamline grid), locating stagnation point of the leading edge is problematic issue. Sometimes the grid quality near the leading edge get worse and convergence problems may arise. Also the stability of the solver has greatly depend on some factors like artificial compressibility factor. The time marching method does not face these kinds of stability problems. Furthermore, choking information must be known by the user if the Newton solver is to be used. On the other hand, there is no such problem for the time marching solver. Final observation is related to the future work of the methods. The Newton solver treats each horizontal gridline as a streamline. Therefore, the viscous flow modelling can only be made by solving Von Karman integral momentum equations. This restricts the possible future works for Newton solver. However, the time marching method is suitable for any kind of viscous modelling. The viscous flow can be modelled not only by the integral momentum equations but also by solving laminar and turbulent stresses within control volume. There is a huge difference between the methods in terms of computation time. For a moderate grid size (86 x 17), the Newton method solves the domain in two seconds. This time reaches about two minutes for the time marching solver. If finer grid is required for a case, the computation time for time marching method may reach about five minutes. Due to this drawback time marching solver may be seen unpractical but the speed of the methods can be increased implementing some acceleration techniques like multigrid acceleration.

References

AGARD, Working Group 18, (1990) “Test Cases for Computation of Internal Flows in Aero Engine Components” AGARD AR 275

Bilgiç, M. (2015), “Development of a Blade to Blade Solver for Axial Turbomachinery”, M.Sc. Thesis, Department of Mechanical Engineering, Middle East Technical University.

Blazek, J. (2001), “Computational Fluid Dynamics: Principles and Applications”, 1st edition, Elsevier.

Drela, M. (2008), “A User’s Guide to MISES 2.63”.

Dunker, D.J. (1990), “Test Case E/CO-4 Single Transonic Compressor Stage”, AGARD AR-275,245-285.

Giles M.B. (1985), “Newton Solution of Steady Two-Dimensional Transonic Flow”, Ph.D. Thesis, Department of Aeronautics and Astronautics, Massachusetts Institute of Technology.

Hirsh, C. (1988), “Numerical Computation of Internal and External Flows- Volume 1&2”, John Wiley & Sons.

Thomas, L.H. (1949), Elliptic Problems in Linear Differential Equations over a Network, Watson Sci. Comput. Lab Report, Columbia University, New York.

Uzol O. (1995), “Quasi-Three-Dimensional Numerical Design of Turbomachinery Blades”, M.Sc. Thesis, Department of Aeronautical Engineering, Middle East Technical University.

Wu, C.H. (1952), “A General Theory of Three Dimensional Flow in Subsonic and Supersonic Turbomachines of Axial, Radial and Mixed Flow Types”, Transactions of ASME, 1363 – 1378.