Validation of a CFD Static Pressure Distribution against Experimental Data for a Turbine Blade

** J. de Villiers, S. Govender*

To achieve numerical flow field results on a particular blade geometry, a 2-dimensional computational flow model of the Steady State Supersonic Cascade at the University of Natal was developed. The geometric model was taken as a 2-D slice through the midspan of the cascade and analysed by FLUENT a commercial Computational Fluid Dynamics (CFD) computer package. The results of this model were then compared with the experimental data points acquired from the static pressure measurements along the midspan of the turbine blade in the experimental cascade. The agreement between the numerical and experimental results is excellent.

Keywords

CFD, FLUENT, experimental, pressure, cascade, turbine, model, flow field, blades

Introduction

The constant need to improve power and efficiency in gas turbine engines leads to the ever-increasing need to raise the combustor exit temperature, and therefore the turbine inlet temperature. However, there is a limit to the allowable temperatures due to the failure of the turbine blades at high temperatures. Many methods exist to prevent the turbine blades from failing, but one of the more generally accepted approaches is to bleed air from the final stages of the compressor section, and use this cooler air to cool the turbine blades via small ducts and serpentine passages within the actual blades. Kennon and Dulikravich,[1] give a method for designing internally cooled turbine blades. Another technique is to film cool the blades by injecting air into the main flow stream from holes and slots in blade, thus creating a shroud of cooler air that protects the blade from the much hotter combustor gases by reducing the gas temperature near the surface of the blade. These methods allow the blade to operate at higher free stream temperatures and thus allows the combustor exit temperature to be raised. Kost and Nicklas,[2] discuss aerodynamic measurements with regards to film cooling while Goldstein and Yoshida,[3] discuss the influence of laminar boundary layers on film cooling performance. In order for the blade to be designed properly or for an existing blade's life to be predicted correctly, the surface heat transfer and flow field conditions must be known. An excellent source of literature is Dunn,[4] who gives an in-depth review on prediction and measurement on heat transfer and aerodynamics in turbines. The University of Natal has a Steady State Supersonic Cascade Rig where flow characteristics and heat transfer coefficients can be measured. Many sets of published experimental results exist for the aerodynamic characteristics and heat transfer coefficients, and are obtainable in literature such as Graziani et al[5], and Bellows and Mayle,[6] amongst others. A large number of references also exist for predicting the aerodynamic and heat transfer characteristics using computational methods. Authors such as Birch,[7] and Ameri et al,[8] discuss Navier Stokes analysis for heat transfer and pressure distributions, while Wang et al,[9] and Rodi and Scheuerer,[10] discuss flows involving low Reynolds numbers. These authors among others use various in house boundary layer codes such as STAN5, GENMIX and ALFA as well as other Navier Stokes methods in conjunction with specific adaptions to solve various cases for heat transfer and pressure distributions. The aim of the current work is to validate a commercially available CFD package FLUENT, which uses various turbulence models, against experimental data for a turbine blade. In general if a computer model can be shown to compare favourably with the experimental results of a particular rotor geometry, then only the computer model is needed to achieve results for other geometries. This allows a quicker acquisition of information and results, which will ultimately lead to better blade geometries and cooling techniques for future applications.

Experimental rig

The continuously operating supersonic experimental cascade rig at the University of Natal comprises of a 4-blade cascade

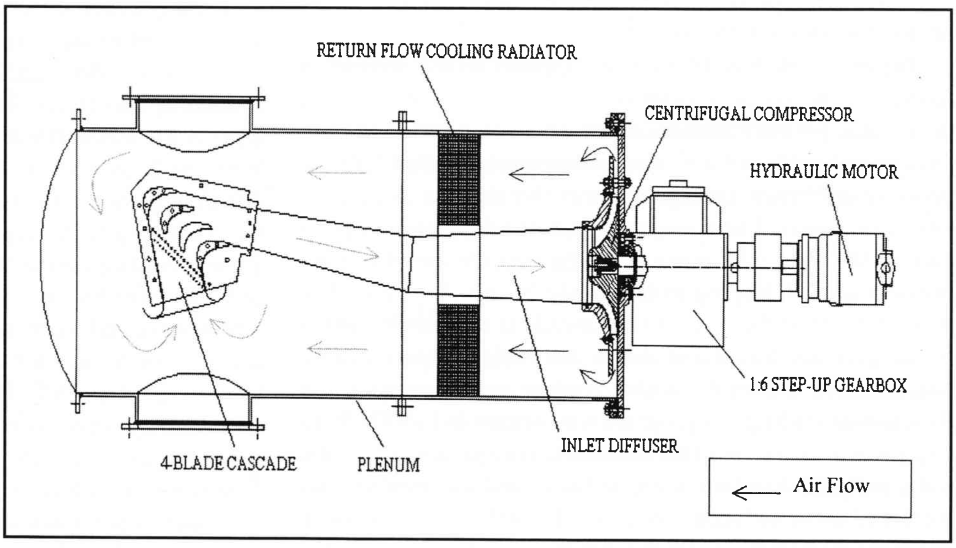

Figure 1: Supersonic cascade rig at University of Natal

* School of Mechanical Engineering, University of Natal, Durban, 4041

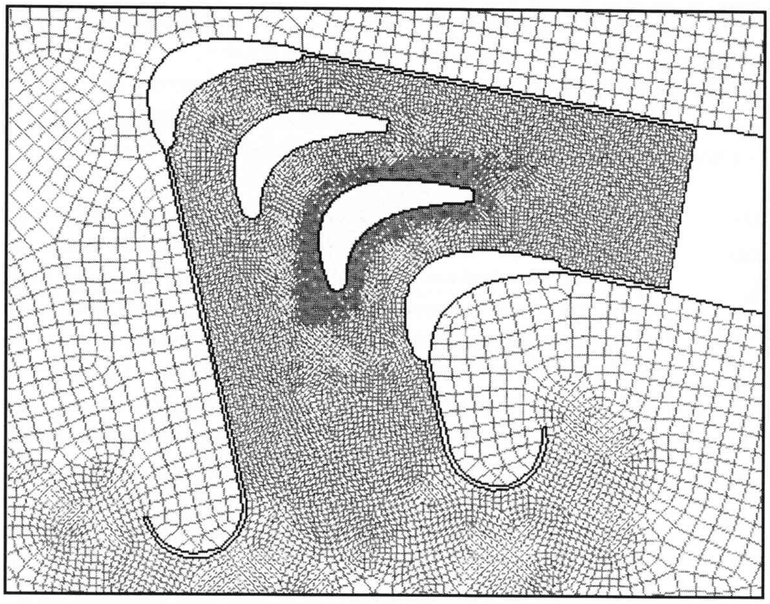

Figure 2a: Close-up view of cascade mesh

housed in a sealed plenum. A compressor rotor, which is driven by a hydraulic motor through a 1:6 step-up gearbox, sucks air through the cascade and expels it back into the plenum (see Figure 1). The temperature of the hot air exiting the compressor is controlled during its return flow to the cascade by passing through an annular water-cooled radiator. This provides realistic engine conditions of $T_{freestrem}/T_{wall}$ = 0.85 for the measurements. The cascade exit diffuser is bent to reduce the effect that the plenum wall would cause on the air flow. The plenum is operated under a vacuum to reduce the power required to run the compressor. The pressure that the plenum is sucked down to can be varied but for the test case it was set at 0.4 bar. The pressure is controlled using the cascade total inlet pressure as an input to a Schimdt trigger that opens or closes a solenoid valve on the input pipe to the continuously running vacuum pump. The speed of the compressor rotor is also controlled and can be altered from 0 RPM to 3000 RPM, but the seal on the compressor rotor shaft is only effective above 200 RPM. Altering the speed of the compressor controls the Mach number while altering the total plenum pressure controls the Reynolds number of the flow through the cascade.

The test blade has 48 pressure tappings drilled around its midspan. The pressure readings are taken by a Rosemount differential pressure transducer, which was calibrated to have an output of 1.8 V to 9.4 V corresponding to a 0 Pa to 40 000 Pa pressure difference, through a Scanivalve that has two 1-pole 24-throw wafers. The transducer reads the difference in pressure of the total inlet pressure to the cascade and the static pressure at each tapping in turn as the Scanivalve steps. The total pressure at the inlet of the cascade is measured using a Kulite pressure transducer, hence the static pressure can be calculated by taking the Kulite reading and subtracting the Rosemount readings. A program was written in LabVIEW to step the Scanivalve, read the transducer voltage, and output the static pressure distribution on the blade, and the input signals were read using an Eagle Electronics PC30 PGL A/D card that has eight differential analogue inputs.

The blade geometry and FLUENT

When a FLUENT model is set up the geometry must first be created in GAMBIT, the pre-processor. A close-up of the meshed geometry that was created in GAMBIT is shown in Figure 2a. One of the advantages of FLUENT is that it was designed to solve totally unstructured meshes like the one shown in Figure 2a. FLUENT is an unstructured solver because it uses internal data structures to assign an order to the cells, faces and grid points in the mesh and does not force an overall structure to the grid. It does not, therefore, require i,j,k indexing to locate neighbouring cells, as the internal data structures maintain contact between adjacent cells. It was realised that the mesh on the test blade would have to be refined while in the cascade and plenum more course grids would be adequate. A boundary layer mesh was generated on the test blade only to reduce the number of computational cells and a total of 19877 were used. The boundary layer mesh consists of a very small first row size of $a = 0.01$, a growth factor of $b/a = 1.35$ and the number of rows was set to 16. This resulted in a boundary layer mesh thickness of 3.45 mm. This boundary layer size was chosen to provide a value of $y^+ \approx 1$, as described by the choice of near wall modelling treatment in the FLUENT online manual[11]. If the mesh size was reduced further no discernable difference in the results could be seen.

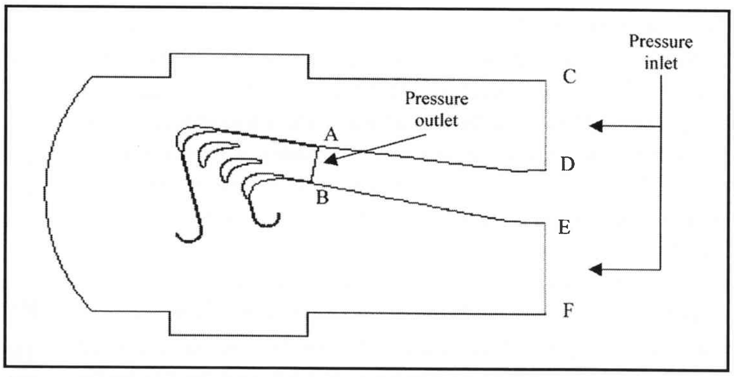

Figure 2b: View of model geometry showing boundaries

The k-ε model, in its realizable form, was used because of its superior performance for flows with separation, complex secondary flow features and where the flow features include strong streamline curvatures. The near wall modelling approach (2-layer zonal method) was used to model the near wall region. With the 2-layer zonal model the flow is separated into two distinct regions, i.e. the bulk flow region (main flow stream) and the viscosity affected region (boundary layer). The separation of these zones is defined by the Reynolds number based on the perpendicular distance (y) from the wall. If the value of $Re_y > 200$ then that cell is considered to be part of the bulk flow region and thus the normal k-ε equations are used to solve the flow. However, if $Re_y < 200$ then the flow is considered to be part of the viscosity affected near wall region and the normal k-ε equations are used except for the turbulent viscosity and dissipation rate which are adapted as suggested by Wolfstein[12].

A turbulence intensity was set as 3% for a free inlet as described by Snedden[13]. A pressure outlet condition was set at the exit of the cascade with an outlet static pressure of 27 kPa, whilst the inlet total and static pressures were set to 40 kPa and 36.5 kPa respectively, as measured in the experiments. The position of the pressure inlet and outlet boundaries are shown in Figure 2b.

The convergence was monitored using the scaled residuals of the conserved variables. The solution was deemed to be converged when the residuals of the conserved variables no longer varied. The FLUENT results are shown in Figure 3. It is noted from Figure 3 that the agreement between FLUENT and the experimental results is excellent for both the pressure and suction surfaces, with some discrepancy on the pressure surface just after the leading edge. The deviations are nonetheless within a tolerance of 5 %.

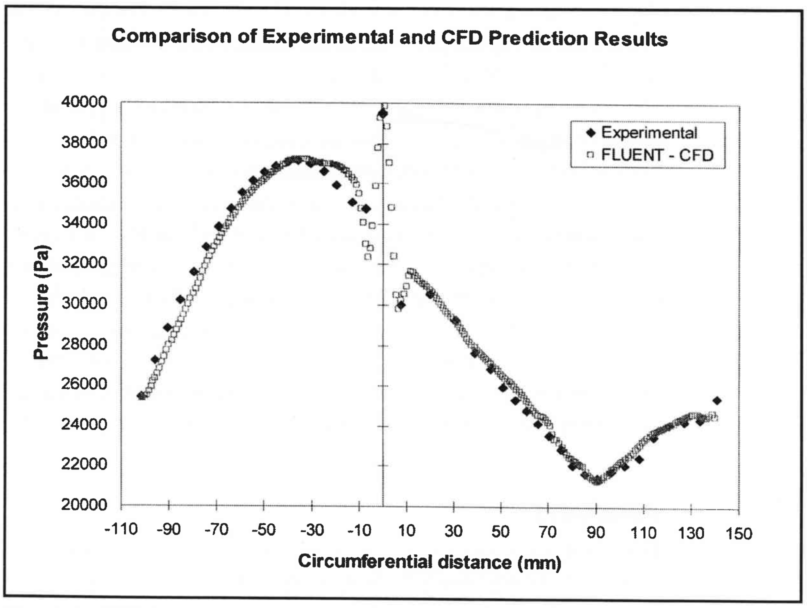

Figure 3: CFD k-e 2-Layer Zonal Method results compared with the experimental results

When analysing the results the main phenomenon that was noted on the FLUENT results, and not found on the experimental results, was the sudden increase in pressure after the characteristic drop of pressure just after the leading edge on the suction surface. This phenomenon was identified as a leading edge separation bubble. The flow reattaches and continues with normal flow through the blade passage. The drastic rise in pressure where the flow is expected to increase in velocity and therefore decrease in static pressure suggests that separation is occurring and causing a bubble that causes a higher pressure on the blade surface. The reason for the formation of the leading edge bubble is caused by the geometry of the blade where the highly curved leading edge suddenly changes curvature to a flat section. This causes the flow to separate from the blade, cause a separation bubble, and then reattach further along the blade. This phenomenon cannot be seen in the experimental graph as the resolution of the pressure tappings is not fine enough due to physical constraints.

Conclusion

It was discovered that the FLUENT results correlate well with the experimentally measured values along the entire circumference of the blade. The main deviation of the numerical results from the experimental data occurs just after the leading edge on the pressure surface, but the numerical values are still within 5% of the experimental results. Further work on the cascade will be done in the future and these results show that FLUENT can be used to good affect to predict the aerodynamic characteristics on the turbine blades. This good correlation shows that quick and easy models can be created in FLUENT to show the pressure distributions on a blade in a cascade, as compared to the manufacturing of an expensive cascade with expensive instrumentation.

References

1. *Kennon, S.R. and G.S. Dulikravich. 1985. "The Inverse Design of Internally Cooled Turbine Blades,"* ASME Journal of Eng. for Gas Turbines and Power, 107, *123-126.*

2. *Kost, F. and M. Nicklas. 2001. "Film-cooled turbine endwall in a transonic flow field: Part 1 –Aerodynamic Measurements,"* ASME TURBO EXPO, *2001-GT-0145.*

3. *Goldstein, R.J. and T. Yoshida. 1982. "The Influence of a Laminar Boundary Layer and Laminar Injection on Film Cooling Performance,"* Journal of Heat Transfer, *104, 355-362.*

4. *Dunn, M.G. 2001. "Convective Heat Transfer and Aerodynamics in Axial Flow Turbines,"* ASME TURBO EXPO, *2001-GT-0506.*

5. *Graziani, R.A., M.F. Blair, J.R. Taylor and R.E. Mayle. 1980. "An Experimental Study of Endwall and Airfoil Surface Heat Transfer in a Large Scale Turbine Blade Cascade,"* Journal of Engineering and Power, *102, 257-167.*

6. *Bellows, W.J. and R.E. Mayle. 1986. " Heat Transfer Downstream of a Leading Edge Bubble,"* ASME Journal, *86-GT-59.*

7. *Birch, N.T. 1987. "Navier-Stokes Predictions of Transition, Loss and Heat Transfer in a Turbine Cascade,"* ASME Gas Turbine Conference and Exhibition, *87-GT-22.*

8. *Ameri, A.A., P.M. Sockol and R.S.R. Gorla. 1992. "Navier-Stokes Analysis of Turbomach-inery Blade External Heat Transfer,"* Journal of Propulsion and Power, *8, 374-381.*

9. *Wang, J.H., H.F. Jen and E.O. Hartel. 1985. "Airfoil Heat Transfer Calculation Using a Low Reynolds Number Version of a Two-Equation Turbulence Model,"* ASME Journal of Eng. for Gas Turbines and Power, *107, 60-67.*

10. *Rodi, W. and G. Scheuerer. 1985. "Calculation of Heat Transfer to Convection-Cooled Gas Turbine Blades,"* ASME Journal of Eng. for Gas Turbines and Power, *107, 620-627.*

11. *Fluent Inc. 1999. "Fluent Inc. Product Documentation,"* root:\Fluent.inc\documentaion\Fluent.inc\manuals\index.html.

12. *Wolfstein, M. 1969. "The velocity and temperature distribution of one-dimensional flow with turbulence augmentation and pressure gradient,"* Int. J.. Heat Mass Transfer, *12, 301-318.*

13. *Snedden, G.C. 1995. "Transient measurement of heat transfer in steady state turbine cascades", Masters thesis, School of Mechanical Engineering, University of Natal, Durban, South Africa.*