ICAST 2000 edited by Wan Ramli Wan Daud et. al

DEVELOPMENT OF A VISCOUS FLOW SOLVER FOR CONDENSING FLOW OF STEAM IN TURBINE

Mohd. Zamri Yusoff[1], Zulkifli Ahmad[1], Rahmat Mohsin[2] and Zainul Asri Mamat[3]
[1]Department of Mechanical Engineering, College of Engineering, Universiti Tenaga Nasional , 43009, Kajang, Selangor, MALAYSIA
[2]Faculty of Chemical and Natural Resources Engineering, Universiti Teknologi Malaysia, Skudai, Johor, MALAYSIA
[3]Power Plant Technology Unit, TNB Research Sdn. Bhd., Bangi, Selangor, MALAYSIA

Key words : Steam, Turbine, Viscous, Turbomachinery, CFD

ABSTRACT

It has been generally accepted that the presence of two-phase flows in the low pressure (LP) stages of steam turbines, due to condensation, has caused extra loss in the stages. This loss is due to thermodynamics irreversibility associated with the spontaneous condensation. However, due to the complexity of the flow in turbomachinery the phenomena involved is insufficiently understood. Theoretical treatments are available for investigating the phenomena concerns with the development of reliable computational tool for flow field predictions. This paper discuss the development of a viscous flow solver for condensing flow of steam in turbines. As a starting point an independent boundary layer calculations are used. This is integrated into the inviscid Euler solver via the viscous-inviscid interaction method. The comparisons with the experimental data has shown that the boundary layer calculation is able to accurately predict the loss.

INTRODUCTION

Currently, over 80 % of the world energy demands are supplied in the form of electrical energy and with few exceptions, the electrical generators are driven by steam turbines. Despite this central position, little attention appears to have been devoted to the study of the problems associated with the operation and design of these machines in comparison with that given to other prime movers.

In the course of the expansion of steam in a turbine, the state path crosses the saturation line and the fluid first supercools and then nucleates to become a two-phase mixture. The subsequent two-phase flow of steam through the final stages of turbines poses a number of problems which are not encountered in other turbomachines. In conventional power plant, the wetness levels in the last few stages of the LP turbines can be as high as 10 - 12 %. It is agreed in the literature that the nucleating and wet stages of steam turbines are less efficient than those running with superheated steam. This reduction in efficiency is attributed collectively to wetness loss, but the mechanisms which give rise to them are insufficiently understood.

Many research efforts, both experimental and theoretical, have been done in order to improve the efficiency of LP turbine running with wet steam e.g. Zamri (1997), Mamat (1996), White and Young (1994). The theoretical work solves the physical governing equations numerically using computer in order to simulate the real flow process in steam turbine, while the experimental data provide means for validating the theoretical predictions. These are aimed at understanding the mechanisms that cause the extra losses and use the knowledge to design better turbine blade profiles.

Flow in real turbomachines is very complex being three-dimensional, unsteady, viscous and turbulent. However, in many cases, considerable physical insights into the phenomena involved can be gained by simplifying the flow. Wu (1952) showed that, the general three-dimensional flow within a turbomachine can be reasonably described by a combination of two-dimensional solutions in two different planes. The first family is referred to as S1 or blade-to-blade plane and the second one as S2 or meridional plane. The solutions on S2 plane provide information about the mean flow through the turbine, while S1 solutions give information about flow behaviour round individual blade sections. Instead of solving the full three-dimensional equations, considerable saving can be gained by solving the two-dimensional equations and iterate between the two planes.

A two-dimensional blade-to-blade, computer program for two-phase condensing flow of steam has been recently developed [Zamri (1997 & 1998), Zamri et al. (1998)]. The program solves the two-dimensional time-dependent Euler equations using second-order accurate cell-vertex finite volume integration together with the fourth order Runge-Kutta time stepping. To take into account of the two-phase effect, the governing equations are coupled with the equations describing the droplet formation and growth. The applications of the solver to condensing flow of steam in cascade of turbine bladings have shown very good agreement with the experimental data in term of condensation loss and overall blade pressure distributions. However, the program cannot predict the extra losses due to viscous effect which has been found experimentally to be significant in transonic conditions [Mamat (1996)].

As a first step towards developing a fully viscous solver, an independent boundary layer calculations subroutine is used. This program is coupled with the Euler solver by using the viscous-inviscid integral boundary layer interaction method. In this paper the numerical formulations which are used to solve the governing equations will be discussed. This will be follwed by the discussion on the Integral viscous-inviscid integral boundary layer method. Finally the program, will be applied to condensing flows in cascade of turbine blades and the results will be compared with the experimental measurements.

NUMERICAL FORMULATIONS

The governing equations employed are the time dependent Euler equations. The main flow governing equations cast in the finite volume formulation in x-y cartesian coordinates system is : -

$$\Omega \frac{\partial \underline{w}}{\partial t} = -\oint_S (\underline{F}dy - \underline{G}dx) \tag{1}$$

where

$$\underline{w} = \begin{vmatrix} \rho \\ \rho V_x \\ \rho V_y \\ \rho E_0 \end{vmatrix} \qquad \underline{F} = \begin{vmatrix} \rho V_x \\ \rho V_x^{\ 2} + P \\ \rho V_x V_y \\ \rho V_x H_0 \end{vmatrix} \qquad \underline{G} = \begin{vmatrix} \rho V_y \\ \rho V_y V_x \\ \rho V_y^{\ 2} + P \\ \rho V_y H_0 \end{vmatrix} \tag{2}$$

Ω is the volume of the small element with perimeter S as shown in FIGURE 1. The small element shown is one of the finite volumes which are formed by the intersection of quasi-streamlines and quasi-orthogonal lines. The flow variables are stored at the cell vertices. In this paper, only a brief description of the numerical formulations will be given. Detail descriptions can be obtained in Zamri (1997 and 1998). The above equations are solved simultaneously for each finite cell volume using a time marching method.

The spatial integration is done using central discretization which is of second order accuracy. A blend of second and fourth order artificial dissipations with pressure switch are added to the residuals prior to the time integration to remove wiggles from the solution. The temporal integration is done using the fourth order accurate, 4 stage Runge Kutta time stepping. To speed up the convergence, 3 types of convergence acceleration schemes are employed namely, local time-stepping, enthalpy damping and implicit residual averaging.

At inlet boundary, the total pressure, total temperature and flow angle are fixed while the static pressure is extrapolated from the interior. At exit, if the exit flow is subsonic, only the static pressure is fixed, while total pressure, total temperature and flow angle are extrapolated from the interior. If the exit flow is supersonic, all four variables are extrapolated from the interior. The periodicity condition on the bounding streamlines, upstream and downstream of the blade row, is easily satisfied by treating the calculating points on each of the bounding streamline as if they were interior ones. At the solid boundary, normal fluxes are set to zero.

To apply the above conservation equations and solution procedures to the two phase flows, the equations have to be combined with the equations describing the droplet formation and growth and solved simultaneously. The equation for describing the rate of nucleation of the droplet per unit volume is

$$J_{st} = \frac{1}{(1+v)} q \sqrt{\frac{2\sigma_r}{\pi m^3}} \frac{\rho_s(T_G)\rho_G}{\rho_L} \exp\left(\frac{-\Delta G^*}{kT_G}\right) \quad (3)$$

The equation for the rate of growth of the droplet is given by

$$\frac{dr}{dt} = \frac{\lambda}{\rho_L L}\left(\frac{T_L - T_G}{r + 1.59\bar{l}}\right) \quad (4)$$

The two sequences of calculations are carried out separately, but it is essential that the coupling between them be exact. This is achieved by the introduction of the wetness fraction, *w*, into the expression for mixture enthalpy *h* and density ρ to yield : -

$$h = (1-w)h_G + wh_L \quad (5)$$

and

$$\frac{1}{\rho} = (1-w)\frac{1}{\rho_G} + w\frac{1}{\rho_L} \quad (6)$$

The wetness fraction, *w*, in turn may be expressed as : -

$$w = \frac{4}{3}\pi \bar{r}^3 \overline{N} \rho_L \quad (7)$$

where $\overline{N}$ is the number of droplet per unit mass of mixture. The additional information necessary is the equation describing the properties of the liquid and vapor phases. The equation of state adopted for the vapor phase is :

$$\frac{P}{\rho_G R T_G} = 1 + B\rho_G \quad (8)$$

where *B* is the second virial coefficient and thermodynamic properties of steam are calculated from mutually consistent relationships. The above systems of equations are sufficient to describe the flow completely.

INTEGRAL BOUNDARY LAYER METHOD

In the viscous-inviscid interaction method the viscous effects are assumed to be concentrated in the boundary layers adjacent to the blade surfaces and the main flow-field is regarded as inviscid and described by the Euler equations. The boundary layer subroutine covers the laminar boundary layers, natural transition and transition through bubble separation, turbulent boundary layers and wakes.

Laminar Boundary Layer

The method of Thwaites (1949) has been adopted due to its simplicity and adequate accuracy. This method consists of the numerical integration of the momentum integral equation, using auxiliary relationships for skin friction and shape factor as functions of pressure gradient parameter, Λ. Thwaites showed that the momentum thickness, θ, is given by : -

$$\Delta\theta^2 = \frac{0.45\nu}{\overline{V}_e^6}\int_0^x V_e^5\,dx \tag{9}$$

where, the bar over V_e denotes the average value over an increment Δx.

In order to calculate the skin friction coefficient and the displacement thickness, the modified expressions for $L(\Lambda)$ and $H(\Lambda)$ as given by Cebeci (1977) are used. δ^* and C_f can be evaluated as : -

$$\delta^* = H\theta \tag{10}$$

$$C_f = \frac{\tau_{wall}}{\frac{1}{2}\rho_e V_e^2} = \frac{2\nu}{V_e^2}\left(\frac{\partial V}{\partial y}\right)_{y=0} = \frac{2\nu}{V_e\theta}L(\Lambda) \tag{11}$$

Laminar-Turbulent Transition

Boundary layer transition from laminar to turbulent flow can occur either by natural transition or laminar separation bubble ending in reattachment as a turbulent boundary layer.

In natural transition, there is a gradual increase in the proportion of the flow which is turbulent at any instant (intermitency) from zero to one. Natural transition is assumed to occur when the momentum thickness Reynolds number reaches a critical value, which is a function of the pressure gradient parameter and the turbulence level.

In a laminar separation bubble, the laminar boundary layer separates and forms a laminar free shear layer which eventually undergoes transition to turbulence. The turbulent free shear layer gains sufficiently high energy fluid from the free-stream by diffusion to reattach as a turbulent boundary layer. Bubble transition occurs if the pressure gradient parameter, Λ , falls below -0.09. The bubble length and the condition on reattachment are determined by two empirical correlations by Horton (1968). The flow was assumed to undergo transition to turbulence at the start of bubble separation. It is also assumed that boundary layer transition occurs instantaneously upon interaction with a shockwave.

Turbulent Boundary Layer

The lag-entrainment method originated by Head (1958) is used in the treatment of turbulent boundary layers. This method employs three differential equations; the momentum integral equation, the entrainment equation and an equation describing the streamwise rate of change of entrainment coefficient. The first two equations are those used in Head's original treatment, the last one is an equation for shear stress developed by Bradshaw et al. (1967). The method also allows a first-order approximation for compressibility. The three equations are as follows : -

$$\frac{d\theta}{dx} = \frac{C_f}{2} - \left(H + 2 - Ma_e^2\right)\frac{\theta}{V_e}\frac{dV_e}{dx} \tag{12}$$

$$\theta\frac{d\overline{H}}{dx} = \frac{d\overline{H}}{dH_1}\left\{C_E - H_1\left(\frac{C_f}{2} - \left(H+1\right)\frac{\theta}{V_e}\frac{dV_e}{dx}\right)\right\} \tag{13}$$

$$\theta\frac{dC_E}{dx} = F\left\{\frac{2.8}{H+H_1}\left[\left(C_\tau\right)_{EQ0}^{\frac{1}{2}} - \lambda_b C_\tau^{\frac{1}{2}}\right] + \left(\frac{\theta}{V_e}\frac{dV_e}{dx}\right)_{EQ} - \frac{\theta}{V_e}\frac{dV_e}{dx}\left[1 + 0.075Ma_e^2\frac{1+0.2Ma_e^2}{1+0.1Ma_e^2}\right]\right\} \tag{14}$$

where, Ma_e is free-stream Mach number, C_τ is the shear stress coefficient, λ_b is the overall scaling factor and suffices EQ and 0 indicate the values in equilibrium flow and zero pressure gradient respectively.

The auxiliary relations for other boundary layer parameters; C_f, H, H_1, $d\overline{H}/dH_1$, C_τ and F in the above equations are that given by Green et al. (1973). Equilibrium flows are defined as those in which the shape of the velocity and shear stress profiles in the boundary layer do not vary with distance, x. Therefore, throughout the flow, dH/dx and $d(C_\tau)_{\max}/dx$ are both zero. The auxiliary equations for the equilibrium quantities are taken from expressions recommended by Green et al. (1973).

The boundary layer growth is calculated by the simultaneous forward integration of the three differential equations using Runge-Kutta technique for the three independent variables, momentum thickness, θ, transformed shape factor, $\overline{H}$ and the entrainment coefficient, C_E .The distribution of displacement thickness, δ^* and other boundary layer parameters are obtained from these.

Treatment of Wake

The basic equations governing the flow in boundary layers are equally applicable to wakes. Thus, the treatment of attached boundary layers described above can be applied to wakes with only minor modifications. The skin friction, C_f is set to zero and the overall scaling factor, λ_b is halved.

NUMERICAL PROCEDURES

In the viscous-inviscid interaction procedure, the inviscid free-stream flow for a given blade geometry is first determined and then the boundary layer displacement thickness corresponding to the blade inviscid velocity distribution is evaluated. The displacement thickness on both suction and pressure surfaces are then added in order to modify the blade geometry. The process is then repeated until the solutions converge. In the numerical solution, the boundary layer calculation is called at 400th time step and subsequently at an interval of 200 time steps until the solutions converge.

The boundary layer is initially laminar around the leading edge, but the pressure distribution obtained in this region is affected by numerical errors. For this reason, the start of the boundary layer treatment is delayed for a distance equivalent to about 2-3 % of the blade chord. At the starting point, the values of θ and C_f are assumed to be zero and 0.025 respectively. This simplification has little effect on the accuracy because the displacement and momentum thickness are very small in this region.

BLADE PROFILE LOSS AND WETNESS LOSS

The profile loss of a turbine blade is defined as the loss due to boundary layer growth on the blade surface and the subsequent dissipation in the blade wake. The boundary layer loss is calculated by : -

$$Bl_{loss} = 1-\left(1-\frac{\theta}{Pitch-\delta^*}\right)^2 \text{ x } 100 \ \% \qquad (15)$$

where θ and δ^* represent the summation of their values on the suction and pressure sides at the trailing edge plane. The mixing loss is given by : -

$$Mixing_{loss} = \frac{K.E.-\frac{1}{2}\left(\overline{V}_{x\ mix}^{2}+\overline{V}_{y\ mix}^{2}\right)}{K.E.} \text{ x } 100 \ \% \qquad (16)$$

The extra loss incurred due to two-phase effects is called thermodynamic wetness loss. The loss is due to the departure of the fluid from thermodynamic equilibrium. The latent heat of the phase change is given up to the liquid before it is transferred back to the vapour. The process is thermodynamically irreversible. In the case of diffusion of flow or encounter with shockwave, there will have to be some evaporation from the droplets. For this purpose, they will have to receive heat from surrounding which will again occur irreversibly. To estimate the magnitude of these losses a method based on the calculation of entropy increase along the quasi-streamlines has been used. The increase in the specific entropy of the fluid for such transfer of heat may be written as : -

$$\Delta s = \int ds = \int L dw \left(\frac{1}{T_G} - \frac{1}{T_L} \right) \tag{17}$$

where L and w are latent heat and wetness fraction respectively. By integrating the above expression along each the quasi-streamline up to the exit plane and taking the pitch average, the loss can be calculated.

RESULTS

The flow solver developed was applied to condensing flows of steam over cascade of a nozzle profile. The general shape of the profile is shown in FIGURE 2. Experimental measurements have been performed by Mamat (1996) in the blow-down tunnel at the University of Birmingham, U.K. The calculations were performed for various flow conditions similar to the experiments covering subsonic, sonic and supersonic dry and nucleating flows. FIGURE 3 shows the mesh arrangements used. A typical flow field predictions (contours of constant density) for supersonic condition for dry and nucleating flows are shown in FIGURES 4 and 5 respectively. Also shown in the figures are the experimentally observed Mach-Zhender photographs for both cases. It can be seen that for both cases the flow features have been predicted accurately. Comparing the dry and nucleating flow, it can be observed that the features are similar except in the throat region. In this region the trailing edge shockwaves systems are weaker for the nucleating flow. The reason for weaker shockwave is condensation that occurred in the throat region. This is shown clearly in the contours of constant wetness fraction as shown in FIGURE 6. Nucleation seems to be started around the throat of the passage and the associated release of latent heat to the vapor caused the flow to decelerate and hence producing weaker shockwave systems. In FIGURES 7 and 8 the predicted values of blade surfaces and mid-passage pressure distributions are compared with the experimental data for the dry and nucleating flows respectively. It can be seen that the agreements between them are excellent. The nucleation region signified by the knee shape in the suction surface pressure distribution have been predicted accurately.

The overall accuracy of the boundary layer calculations can be determined by comparing the measured and calculated expansion efficiency. The predicted total loss is the summation of wetness, boundary layer, mixing and shock losses. The overall results obtained from all the calculations are compiled in TABLE 1. It can be seen that the predicted expansion efficiencies are in very close agreement with the experimental data. It can be concluded that the boundary layer calculation is able to accurately predict the loss accurately.

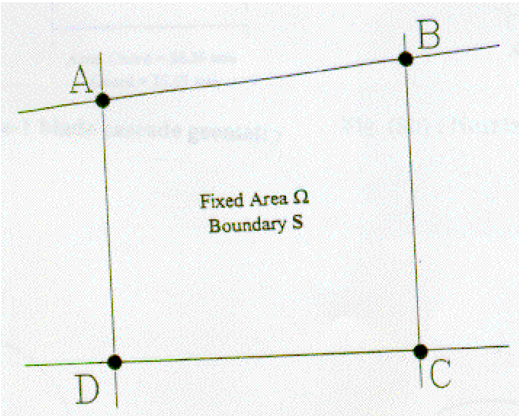

FIGURE 1 : An Element of A Mesh

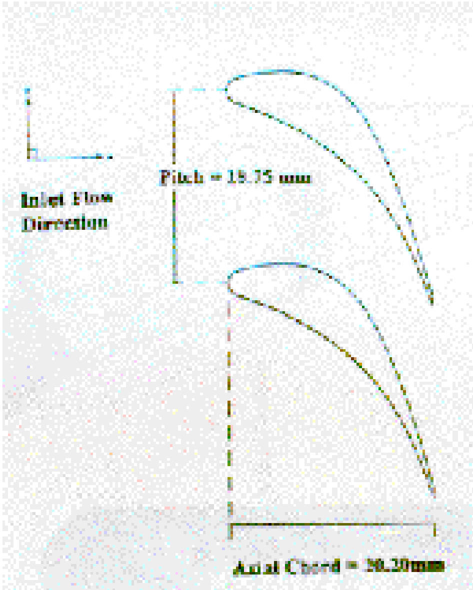

FIGURE 2 : Nozzle Blade Cascade Geometry

CONCLUSIONS AND FUTURE WORKS

The development of the viscous flow solver for condensing flow of steam in turbine is described. As a starting point an independent boundary layer calculations are used. This is integrated with the inviscid Euler solver with the viscous-inviscid interaction method. The comparisons with the experimental data has shown that the boundary layer calculation is able to accurately predict the loss. However, because of the nature of the viscous-inviscid interaction method, it is not possible to predict the direct interaction between the boundary layer and main flow field such as shock boundary layer interaction. In order to predict this, direct inclusion of the viscous term including the turbulence terms into the governing equations are needed. The next step of the work is the inclusion of these terms into the governing equations. This would also open the possibility of investigating the condensations in the boundary layer region.

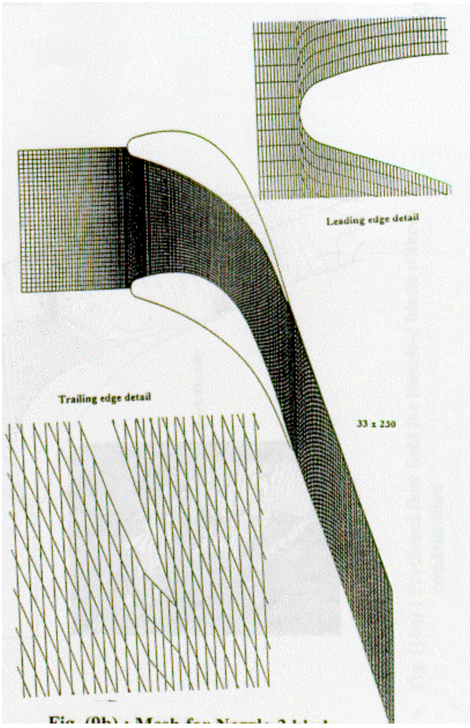

FIGURE 3 : Mesh Arrangement

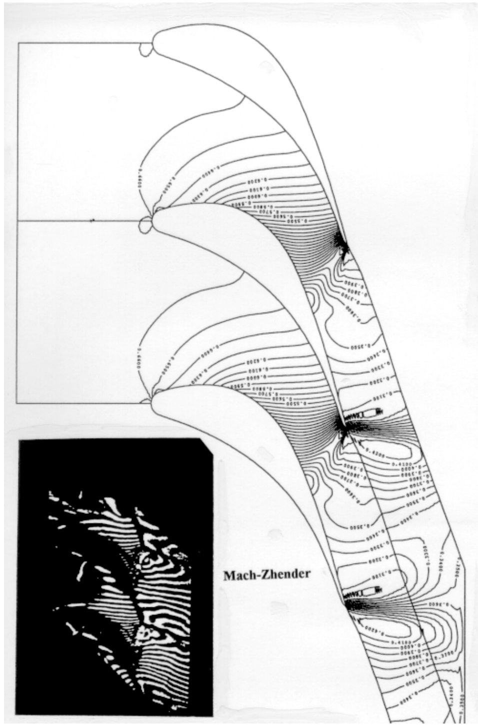

FIGURE 4 : Constant Density Contours – Dry Flow

NOTATIONS

E_o Total internal energy
F Scaling function in skin friction law
G Gibbs free energy
H Mean velocity shape factor
H_1 Mass flow shape factor
H_o Stagnation enthalpy
h Specific enthalpy
J Rate of formation of critical nuclei per mass per unit time
k Boltzman's constant
K.E. Kinetic energy
L Latent heat (h_g - h_L) or Thwaites parameter
m Mass of molecule
P Static pressure
q Condensation coefficient
R Gas constant of vapour

r	Droplet radius
s	Specific entropy
T	Temperature
t	Time
δt	Time step for the droplet growth calculation
V	Overall velocity
ρ	Density
λ	Thermal conductivity
Λ	Thwaites pressure gradient parameter
τ	Shear stress
ν	Kinematics viscosity, specific volume or correction factor in nucleation rate equation
δ	Overall boundary layer thickness, incremental change
δ^*	Displacement thickness
σ	Surface tension

Subscript

G	Vapour phase
L	Liquid phase
max	maximum
mix	mixed values
r	Droplet
s	Saturation
st	Steady state

REFERENCES

Zamri, M.Y. *An improved treatment of Two-Dimensional Two-Phase Flows of Steam by a Runge-Kutta Method*', 1997, Ph.D. Thesis, Department of Manufacturing and Mechanical Engineering, The University of Birmingham, U.K.

Mamat, Z.A. 1996 *The Performance of a Cascade of Nozzle Turbine Blading in Nucleating Steam*, Ph.D. Thesis, Department of Manufacturing and Mechanical Engineering, The University of Birmingham, U.K.

White, A.J. and Young, J.B. 1994, Time Marching Method for the Prediction of Two-Dimensional, Unsteady Flow of Condensing Steam, *J. Propulsion and Power*, **9** (4).

Wu, Chung-Hua, 1952, *A General Theory of Three-Dimensional Flow in Subsonic and Supersonic Turbomachines of Axial Radial and Mixed-Flow Types* . NASA TN 2604

Mohd. Zamri Yusoff, Ibrahim Hussein, Mohd. Azree Idris, 1998 Prediction and Improvement of Steam Turbine Performance Using Computational Fluid Dynamics (CFD), *Proceedings of the 1st TNB Technical Conference*, Universiti Tenaga Nasional, July, 1998.

Mohd. Zamri Yusoff, 1998, A Two-Dimensional Time-Accurate Euler Solver for Turbo-machinery Applications, *Journal of Institute of Engineers*, Malaysia, **59** (3)..

Thwaites, B.,1949, Approximate Calculation of the Laminar Boundary Layer *Aero. Quarterly*, **1** : 245-280.

Cebeci, T. 1977 , *Calculation of Laminar and Turbulent Boundary Layers for 2-d Time Dependent Flows,* N.A.C.A. C.R., No. 2820.

Horton, H.P. 1968, *Laminar Separation Bubbles in Two and Three Dimensional Incompressible Flow,* P.h.D. Thesis, Queen Mary College, Dept. of Aero. Eng.

Head, M.R. 1958, *Entrainment in Turbulent Boundary Layer*, A.R.C.R. and M. No. 3152, September 1958.

Bradshaw, P., Ferriss, D.H. and Atwell, N.P. 1967, Calculation of Boundary Layer Development Using the Turbulent Energy Equation, J. Fluid Mech., **28** (3) : 593-616

Green, J.E., Weeks, D.J. and Brooman, J.W.F. 1973, *Prediction of Turbulent Boundary Layer and Wakes in Compressible Flow by Lag Entrainment Method,* A.R.C., R. and M., No. 3791.

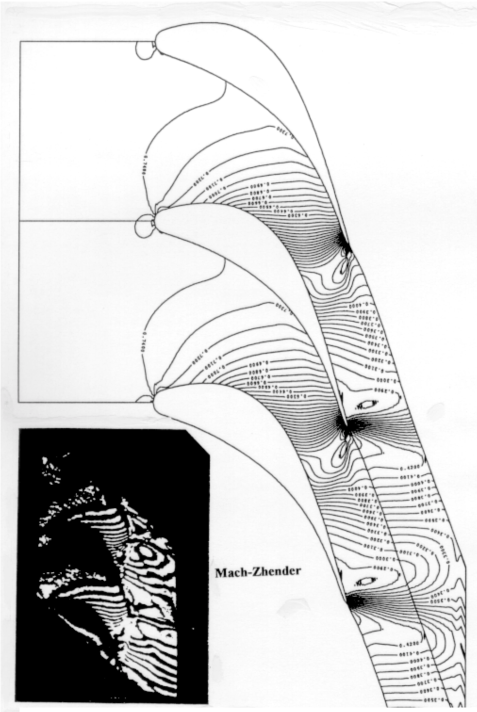

FIGURE 5 : Constant Density Contours – Nucleating Flow

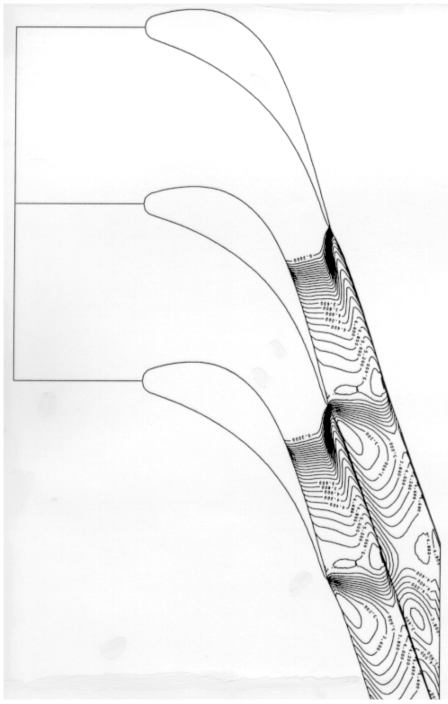

FIGURE 6 : Constant Wetness Fraction Contours – Nucleating Flow

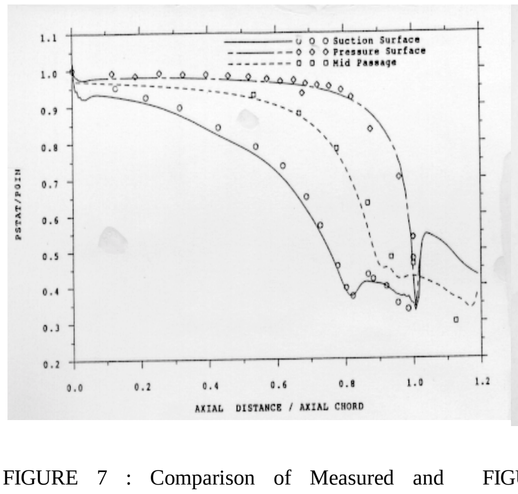

FIGURE 7 : Comparison of Measured and Predicted Surfaces and Mid Passage Static Pressure Distributions – Dry Flow

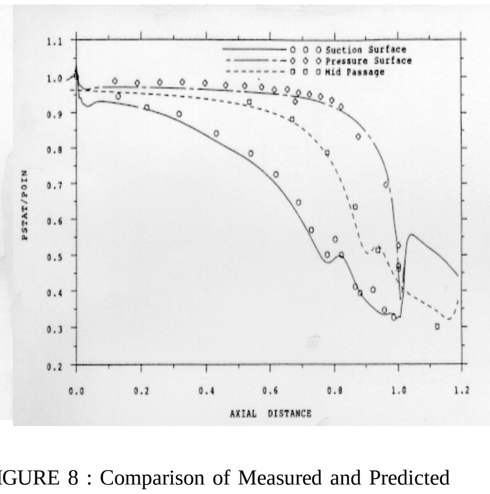

FIGURE 8 : Comparison of Measured and Predicted Surfaces and Mid Passage Static Pressure Distributions – Nucleating Flow