COMPUTATIONAL ANALYSIS OF FLOW THROUGH TURBINE INLET GUIDE VANE’S PASSAGE

By

Sakhr B. Abu Darag
SUST, Engineering College, Aeronautical Engineering Dept., P.O. Box 407, Khartoum, Sudan

ABSTRACT
In this work, comparisons between the results of three Computational Fluid Dynamics (CFD) codes are performed. The computations are made for an inlet guide vane of turbine. The CFD codes used in the computation are in-house codes. The computation of the stage 3D code was done at Siemens, Sweden, whereas the turbulent and laminar computations were done at Chalmers University. H-mesh structure has been used in the computations and $k - \varepsilon$ model has been tested for the two turbulent codes to figure out the influence of the turbulence model. The results indicate that, laminar code predicts clearly the wake behind the vane compared with stage 3D, whereas it is limited in turbulent result. The calculations show that at some points just before the trailing edge from 96% to 98% $C_{ax}$, on the pressure side, the static pressure decreases sharply up to 20 (kPa) for the turbulent and 14 (kPa) for the laminar flow. Also it has been found that the calculated kinetic energy in laminar and $k - \varepsilon$ codes are fairly agreed with stage 3D.

KEYWORDS: Inlet Guide Vane, Computational Fluid Dynamics (CFD), Turbulence Model, Wake, Trailing Edge.

الملخص:
تم في هذا العمل تصابيب ومقارنة نتائج ثلاثة برامج لحوسبة ديناميكا الموائع. استخدمت ريشة مدخل موجهة لتوربين كمودل لإجراء التحصيب، جميع البرامج التي تم استخدامها هي ذاتية الملكية ولم تستخدم تجاريا. تم تنفيذ الحوسبة عن طريق برنامج stage3D بشركة سيمنس بالسويد بينما تم التحصيب ببقية البرامج بجامعة شالمرز، H-mesh للاختبار مع مودل $k - \varepsilon$ للكتاليوجيا. تم استخدام نوع خلف الريشة بصورة واضحة مقارنة wake لعنصر الانسياب المعضور على الموائل. بدأنا نتائج البرنامج الانسياب الطبيب استطاع اظهار ظاهرة $C_{ax}$ وتم استخدام برنامج stage3D بينما كانت محدودة في برنامج الانسياب المعضور. أوضحت النتائج أيضا انخفاض الضغط الانسيابي بصورة حادة على السطح المنضوع للريشة في بعض النقاط قبل نهاية الريشة إلى 20 في حالة (kPa).
الإنساب الضغط و (14 kPa) وجد أيضاً أن طاقة الحركة في الإنساب الطبقي والمضطرب متطابقة لحد كبير مع البرنامج الانموذج stage3D.

**Nomenclature**

**Latin symbols**

- $e$: specific internal energy
- $h$: specific enthalpy
- $k$: kinetic energy, thermal conductivity
- $L$: turbulent length scale
- $p$: pressure
- $P$: production term
- $q$: heat flux
- $R$: specific gas constant
- $s_{ij}$: strain rate tensor
- $t$: time
- $T$: Temperature
- $u_i$: Velocity in $x_i$ direction
- $u,v,w$: Velocity in Cartesian coordinates
- $x,y,z$: Cartesian coordinates

**Greek symbols**

- $\delta_{ij}$: kronecker delta
- $\varepsilon$: dissipation rate
- $\rho$: density
- $\mu$: dynamic viscosity
- $\mu_t$: turbulent dynamic viscosity
- $\tau_{ij}$: shear stress
- $\phi$: arbitrary flow variable

**Superscripts**

- $eff$: effective
- $iso$: isentropic

**INTRODUCTION:**

Serious development of the gas turbine mainly for military purposes began before World War II. After the war, the commercial gas turbine for civil purposes like civil aircraft and for power generation began to compete successfully in the mid last century (Cohen, 1998).

In a simple gas turbine system the compressed air is supplied by the compressor to the combustion chamber. In the combustion chamber the fuel is burned at a high temperature, then expansion takes place in the turbine to atmospheric pressure. In the real gas turbine, the compressor and the turbine units consist of several compressors and turbines, usually referred to stages.

Nowadays, the increase in computation power has made numerical techniques economical and we can obtain results in reasonable time. In addition numerical calculations give complete picture about the flow field, since experiments has a limitation due to availability.
OBJECTIVE

The aim of this work is to validate the laminar and turbulent in-house CFD codes for the turbo machinery applications. The model used in the computation is the Inlet Guide Vane (Nozzle blade) model for loaded Gas Turbine engine represented by figure 1 and 2. The code used for the validation process is the standard stage3D, the one which indicates a very good agreement with the experimental results in master diploma made by Joda (2004).

GOVERNING EQUATIONS

A compressible flow according to Wilcox (1993) is the flow in which significant density changes occurs even when pressure changes are small. The governing equations for compressible flow in Cartesian coordinates are:

Mass conservation equation:
\[
\frac{\partial \rho}{\partial t} + \sum_{j=1}^{3} \frac{\partial (\rho u_j)}{\partial x_j} = 0 \quad \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots
\]
Energy equation:

\[
\frac{\partial}{\partial t} \left[ \rho (e + \frac{1}{2} u_i u_j) \right] + \frac{\partial}{\partial x_j} \left[ \rho u_j (h + \frac{1}{2} u_i u_j) \right] = \frac{\partial}{\partial x_j} (u_j \tau_{ij}) - \frac{\partial q_j}{\partial x_j} \quad \text{......... (3)}
\]

For Newtonian fluid:

\[
\tau_{ij} = 2 \mu s_{ij} - \frac{2}{3} \mu \frac{\partial u_k}{\partial x_k} \delta_{ij} \quad \text{................................................. (4)}
\]

\[
s_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \quad \text{................................................. (5)}
\]

State equation:

For gases, we use the ideal gas law

\[
p = \rho RT \quad \text{................................................. (6)}
\]

Fourier’s assumption reads

\[
q_i = -k \frac{\partial T}{\partial x_j} \quad \text{................................................. (7)}
\]

TURBULENCE MODELING

The most popular two-equation model considered by Lars Davidson (2002) is the \( k - \varepsilon \) model. The model provides these variables by solving two transport equations, one for \( k \) and one for \( \varepsilon \). The turbulent viscosity \( \mu_t \) is modeled as the product of a turbulent velocity \( U_t \) and a turbulent length scale \( L \), as proposed by Prandtl and Kolmogorov.

Introducing a proportionality constant gives:

\[
\mu_t = \rho C_{\mu} U_t \quad \text{................................................. (8)}
\]

Where \( C_{\mu} \) is constant.

The velocity scale \( U_t \) is calculated as square root of the turbulent kinetic energy

\[
U_t = \sqrt{k} \quad \text{................................................. (9)}
\]

In the standard \( k - \varepsilon \) model it is assumed that the dissipation \( \varepsilon \) can be related to large scale, therefore:

\[
L = \frac{k^{3/2}}{\varepsilon} \quad \text{................................................. (10)}
\]
Where $\varepsilon$ is the turbulent dissipation rate.
The eddy viscosity $\mu_t$ is calculated from the product of a turbulent velocity scale and
turbulent length scale as:

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}$$ .................................................. (11)

The values of $k$ and $\varepsilon$ are obtained from the solution of the following transport
equation:

$$\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_j} (\rho u_j k) = \frac{\partial}{\partial x_j} (\mu_{eff,k} \frac{\partial k}{\partial x_j}) + P_k - \rho \varepsilon$$ ...................... (12)

$$\frac{\partial}{\partial t} (\rho \varepsilon) + \frac{\partial}{\partial x_j} (\rho u_j \varepsilon) = \frac{\partial}{\partial x_j} (\mu_{eff,\varepsilon} \frac{\partial \varepsilon}{\partial x_j}) + \frac{\varepsilon}{k} (C_{\varepsilon 1} P_k - \rho C_{\varepsilon 2} \varepsilon)$$ ............. (13)

Where,

$$\mu_{eff,k} = \mu + \frac{\mu_t}{\sigma_k}, \quad \mu_{eff,\varepsilon} = \mu + \frac{\mu_t}{\sigma_\varepsilon}$$ .................................................. (14)

The production rate is given by:

$$P_k = \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \frac{\partial u_i}{\partial x_j},$$ .................................................. (15)

The constants are:

$$C_\mu = 0.09, \quad C_{\varepsilon 1} = 1.44, \quad C_{\varepsilon 2} = 1.92, \quad \sigma_k = 1.0, \quad \sigma_\varepsilon = 1.3$$

NUMERICAL METHODS
The main stages in a CFD solution as mentioned by Fakhrai (2007) are:

- **Pre-processing:** which contains problem formulation, governing equations, boundary conditions and mesh construction.
- **Solving:** transfer the governing equations into numerical solution.
- **Post-processing:** results plot and analysis.

**Grid Generation**
The main part of pre-processing stage is mesh generation. In stage3D solution the grid has been done by Bladegrid generator (Joda, 2004). For the turbulent and laminar computation the g2dmesh software, which made by Lars Eriksson (2004) has been used to generate a single-block (61×31) H-mesh illustrated in figure 3.
The g2dmesh is a 2D multi-block grid generator based on transfinite Interpolation to generate a straight H-mesh. The method is based on a general interpolation concept where unvaried interpolation operators are combined in a certain manner to obtain desirable properties. In the context of 2D grid generation, we only need a few special formulations of this general interpolation method.

The algebraic grid generation procedure used consists of two basic steps:

1. Generation of grid curves
2. Generation of grid (sub) blocks

---

**The Solvers:**

- **Stage3D Code**, Stage3D as described by Joda (2004) is a structured three-dimension Reynolds averaged Navier-Stokes multistage program based on Dawes program (Dawes 1988, 1991). The program is written for compressible flow in turbomachinery. The governing equations are discretized in space using a cell-centered Finite Volume method, with central scheme. To enhance the stability of the code, smoothing has been used. A time marching algorithm is also used, which is essentially explicit with a residual averaging technique applied to improve the stability of the time marching. A multi-step Runge-Kutta method provides the time marching; two-stage, four-stage and five-stage are implemented. To speed up the convergence full multigrid acceleration is applied. The code has the capability to implement several types of turbulence models like Spalart-Allmaras, Baldwin-Lomax model, \(k-e\) and \(k-w\) model. Furthermore the
program has capability to use the perfect, ideal, real gas assumption (MBStag3D 2.6 user manual).

- **Laminar Code**, The flow solver code g2dnav3 made by Ericsson (2004) is based on the Finite-Volume Method (FVM) and is adapted for structured grids of multi-block type (also named block-structured grids). The cell-centered approach is used, i.e. the grid cells formed by the grid nodes are used as control volumes. The convective (inviscid) flux across each cell face is computed by applying the characteristic flux method, with user-defined upwinding and order of accuracy. Extra numerical dissipation, controlled by a sensor based on pressure and/or density gradients, may be added to this flux in order to obtain improved shock handling capability. The viscous flux across each cell face is computed by approximating all necessary velocity gradients and temperature gradients via 'compact' central differences. The 'method of lines' is used, i.e. the finite-volume discretisation process is first applied to obtain a large system of ODE's (Ordinary Differential Equations) and a 3-stage 2nd-order accurate Runge-Kutta time stepping scheme is then applied to advance the system in time. The time stepping is explicit and two modes are possible: time accurate mode and local time stepping mode. In the first case the same time step is applied in all grid cells whereas in the second case the time step varies from cell to cell. In both cases the time step(s) are based on a user-defined CFL number. The local time stepping is only used for accelerating the convergence towards a steady state solution.

- **Turbulent Code**, The flow solver code 'g2dnav3ketm' is the same in construction as g2dnav3, the only one different is that in g2dnav3ketm code The standard two-equation k-epsilon model is used for turbulence closure and wall functions may be applied at solid walls.

**Boundary Conditions**

The boundary conditions for Stage3D mentioned by Darag & Joda (2004) were taken from KTH test rig measurements (Roux 2003), span-wise distributions of total pressure, total temperature, flow angle, turbulence intensity and absolute velocity, are defined as inlet boundary conditions. In the outlet plane the static pressure is defined at the hub. At all solid walls, such as vane surface, hub and shroud, no-slip boundary condition is defined \( u = v = w = 0 \), also all walls are assumed to be adiabatic (no heat transfer). At other boundaries periodic boundary conditions were used. In the 2D in-house codes g2dnav3 and g2dnav3ketm, the boundary conditions are taken at the middle of span-wise. For g2dnav3, the distribution of total pressure and total
enthalpy were used at the inlet. At the outlet the static pressure was taken from Stage3D solution.
For g2dnv3ketm, the distribution of total pressure, turbulence intensity, total enthalpy, kinetic energy and dissipation were used at the inlet. At the outlet the static pressure also was taken from Stage3D solution. Figure 4 shows the boundaries.

Figure 4: Boundary Condition

**Discretization:**
Discretization is the process whereby the governing differential equations are replaced by their discrete counterparts. The differential equations are transformed to algebraic equations, which should correctly approximate the transport properties of the physical process. The method used to solve the partial differential equations numerically is finite volume method.

**Momentum equation:**
The convective, non-linear term must be linearized so that one U is treated as known, and one is solved for. To differ between these two we denote the U which we solve for by φ, so that the U momentum equation reads:

\[
\frac{\partial}{\partial x} (\rho U \phi) + \frac{\partial}{\partial y} (\rho V \phi) = -\frac{\partial P}{\partial x} + \frac{\partial}{\partial x} \left( \mu \frac{\partial \phi}{\partial x} \right) + \frac{\partial}{\partial y} \left( \mu \frac{\partial \phi}{\partial y} \right) \quad \ldots \quad (16)
\]

Discretize the 2D U equation over it is control volume.
The convective term discretization:

\[
\int_{\Delta V_{i,j}} \left[ \frac{\partial}{\partial x} (\rho U \phi) + \frac{\partial}{\partial y} (\rho V \phi) \right] dx dy = \int_{I-1/2}^{I+1/2} \int_{J-1/2}^{J+1/2} \left[ \frac{\partial}{\partial x} (\rho U \phi) + \frac{\partial}{\partial y} (\rho V \phi) \right] dx dy \quad \cdots \quad (17)
\]

\[
= \int_{I-1/2}^{I+1/2} \left[ (\rho U \phi)_{i,j} \right] dy + \int_{J-1/2}^{J+1/2} \left[ (\rho V \phi)_{j,i} \right] dx \quad \cdots \quad (18)
\]

\[
= [(\rho U \phi)_{i,j} - (\rho U \phi)_{i-1,j}] \Delta y + [(\rho V \phi)_{j,i+1/2} - (\rho V \phi)_{j,i-1/2}] \Delta x \quad \cdots \quad (19)
\]

The pressure term discretization:

\[
\int_{\Delta V_{i,j}} \frac{\partial p}{\partial x} dx dy = \int_{I-1/2}^{I+1/2} \int_{J-1/2}^{J+1/2} \frac{\partial p}{\partial x} dx dy \quad \cdots \quad (20)
\]

\[
= \int_{I-1/2}^{I+1/2} \left( p_{i-1,j} - p_{i,j} \right) dy \quad \cdots \quad (21)
\]

\[
= (p_{i+1,j} - p_{i,j}) \Delta y \quad \cdots \quad (22)
\]

The diffusion term discretization:

\[
\int_{\Delta V_{i,j}} \left[ \frac{\partial}{\partial x} \left( \mu \frac{\partial \phi}{\partial x} \right) + \frac{\partial}{\partial y} \left( \mu \frac{\partial \phi}{\partial y} \right) \right] dx dy \quad \cdots \quad (23)
\]

\[
= \int_{J-1/2}^{J+1/2} \left[ \mu \frac{\partial \phi}{\partial x} \right]_{i,j} dy + \int_{I-1/2}^{I+1/2} \left[ \mu \frac{\partial \phi}{\partial y} \right]_{i,J} dx \quad \cdots \quad (24)
\]

\[
= \left[ \mu_{i,j+1/2} \frac{\partial \phi}{\partial x} - \mu_{i,j} \frac{\partial \phi}{\partial x} \right] \Delta y + \left[ \mu_{i+1/2} \frac{\partial \phi}{\partial y} - \mu_{i,j} \frac{\partial \phi}{\partial y} \right] \Delta x \quad \cdots \quad (25)
\]
The discretized U momentum equation can now be written as:

\[ a_{i,j}U_{i,j} = \sum_{nb} a_{nb} U_{nb} + \left( P_{i-1,j} - P_{i,j} \right) \Delta y \] ................................. (26)

\[ \sum_{nb} a_{nb} U_{nb} = a_w U_{i-1,j} + a_x U_{i+1,j} + a_s U_{i,j-1} + a_n U_{i,j+1} \] ................................. (27)

\[ a_{i,j} = a_w + a_x + a_s + a_n \] .................................................. (28)

The continuity equation will be used as an indirect equation for pressure. First the momentums equations are solved, using an old pressure.

**Continuity Equation:**

\[ \frac{\partial}{\partial x} \left( \rho U \right) + \frac{\partial}{\partial y} \left( \rho V \right) = 0 \] ................................. (29)

\[ = \int_{\Delta y} \int_{\Delta x} \left[ \frac{\partial}{\partial x} \left( \rho U \right) + \frac{\partial}{\partial y} \left( \rho V \right) \right] dxdy \] ................................. (30)

\[ = \int_{l+1/2}^{l+1/2} \int_{j+1/2}^{j+1/2} \left[ \frac{\partial}{\partial x} \left( \rho U \right) + \frac{\partial}{\partial y} \left( \rho V \right) \right] dxdy \] ................................. (31)

\[ = \left[ \left( \rho U \right)_{i+1,j} - \left( \rho U \right)_{i,j} \right] \Delta y + \left[ \left( \rho V \right)_{i,j+1} - \left( \rho V \right)_{i,j} \right] \Delta x = 0 \] ................................. (32)

**RESULTS AND DISCUSSION**

The acceleration of the flow through the nozzle between the pressure and suction surfaces in all CFD calculations presented in figure 5 is quite clear and it is referred to the inverse pressure gradient.

The periodicity represents the effect of the vane trailing edge on the flow velocity downstream the passage. This effect represented by some regions whereas the flow velocity is unstable and propagates in all directions.

Laminar code predicts clearly the wake behind the vane compared with stage3D, whereas it is limited in k-eps result.
Figure 6 illustrates the static pressure reduction through vane’s passage in k-eps and laminar calculations, which is very close to stage3D. The high static pressure region is concentrated on the stagnation point of the vane leading edge.

The flow density represented by figure 7 indicates the flow rate reduction through the passage. This reduction in density is inversely proportional to the velocity according to continuity equation.

Figure 8 represents the kinetic energy of the flow. The calculated kinetic energy from laminar and k-eps codes indicates fairly agreement with stage3D.

Figure 9 shows the measured and calculated static pressure distribution on the vane surface at mid-span on the pressure side and suction side. The static pressure is plotted versus the normalized axial chord length \( C_{ax} \).

The calculations show that there are some points just before the trailing edge from 96% to 98% \( C_{ax} \), on the pressure side, where the static pressure decreases sharply up to 20 kPa for the k-eps and 14 kPa for the laminar.

On the pressure side all the calculations predicted the static pressure fairly well from leading edge up to about 92% \( C_{ax} \), while on the suction side there is under prediction up to 55% \( C_{ax} \) for k-eps and 65% for laminar.

Obviously, the agreement between measured and calculated isentropic Mach number on the pressure side is better than on the suction side.

The isentropic Mach number at the tip and mid-span is predicted using Stage3D and the result is illustrated in figure 10 and 11 respectively.

The velocity increased downstream the passage from tip to mid-span due to the no-slip condition assumption in the wall. Furthermore stage3D predicted the wake fairly good at mid-span, whereas it is not observed at the tip endwall.

The gas in the calculations for all codes is considered as ideal gas, which is represented by density distribution.
Figure 5: Flow Mach Number Distribution

Figure 6: Flow Static Pressure Distribution
Figure 7: Flow Density Distribution

Figure 8: Flow Kinetic Energy Distribution
Figure 9: Static Pressure distribution around the vane surface

Figure 10: Isentropic Mach number distribution at the tip endwall.
CONCLUSION
Two dimensional steady compressible flow calculations have been performed in this study for two in-house CFD codes and a comparison has been made with recommended Stag3D code for validation. Straight H-mesh structure has been used and k-eps model is fixed to catch the turbulence. The calculations have been performed for an inlet guide vane of a gas turbine. It is clear that the laminar calculation predicts clearly the wake behind the vane, whereas the effect is limited in turbulent results. Good agreement is found for the calculated kinetic energy in the three CFD codes. It is obvious that the turbulent code is more efficient for turbo machinery predictions than the laminar code. In addition the last one is fairly predicts the viscous flows.
REFERENCES