



Modelling and Simulation of Fluid Flow and Heat Transfer in Rotor-Stator Disc System

A. F. ABBASI, D. M. PATHAN,* A. A. MEMON, M. MEMON, K. HARIJAN

Mehran University of Engineering and Technology Jamshoro, Sindh, Pakistan

Received 11th March 2012 and Revised 4th August 2012

Abstract: The focus of present work is to investigate the fluid flow and heat transfer behaviour in the rotor-stator disc flow system for the range of mass flow rates $2530 \leq C_w \leq 9680$ and a fixed value rotational Reynolds number $Re_\Omega = 1.25 \times 10^6$. The predictions have been obtained through a finite-difference, axisymmetric, steady-state and elliptic method, in the cylindrical polar co-ordinate system. A CFD commercial code Fluent has been implemented by provoking the two turbulence models, the low Reynolds number k- ϵ and the low Reynolds number second moment closure models. The predictions show the basic characteristics of the fluid flow and heat transfer in the rotor-stator system for the range of mass flow rate. For a lower mass flow rate, a strong recirculating core of fluid was produced, which formed the axial-wind between the two discs, consequently heat is transferred from hot rotor disc to air and then to the stator. For a higher mass flow rate a strong wall-jet was predicted along the rotor disc, result of that maximum heat is transferred in the outer region of the cavity. The predicted flow structure, temperature field and the Nusselt numbers show the basic characteristics of the rotor-stator disc flow system. A comparison of the computed Nusselt numbers shows a good level of agreement with the measurements. The predicted results provide the essential information about the windage heating and thermal growth in the rotor-stator disc spaces.

Keywords: Rotor-Stator System, Low Reynolds number Second moment closure.

1. INTRODUCTION

Heat transfer and fluid flow calculations in the rotor-stator disc system is of a practical interest because this system is applicable to the gas turbine engine and the compressor disc spaces. Recent advancement in the turbine entry temperatures and pressure ratios, 1200K to 1900K and 6:1 to 40:1 has amplified the operating conditions of the gas turbine engine. Consequently, the temperature of different components like turbine blades, nozzle guide vanes and rotor-stator discs has been increased and developed the thermal stresses in these components. To increase the endurance life and to obtain the optimum thermal efficiency of the engine is a big issue for gas turbine designers. There are two approaches in practice to subside the overheating issue, advancement in the material composition and the development of an effective internal-air cooling system of a gas turbine engine. The later approach is the best suited for this purpose in which the cooling air is bled from the compressor stages to remove the heat and cool the engine components.

In order to optimize the efficiency and to design an efficient internal-air cooling system many research workers devoted their efforts to this field, experimentally and numerically. In the asymmetrical heating, rotationally-induced buoyancy effects may become significant and the fluid transports from the boundary layer of the hot to the cold disc across the central core region, the so-called axial wind as indicated by (Memon 1995 and Poncet *et al.* 2009). Owen and Rogers 1995)

provides the basic information about the flow structure and the ingress of hot mainstream gas into the rotor and stator disc space. Gan *et al.* 1993) provided the most important features of a rotor-stator system and identified the ingress problem in this system. (Chen *et al.* 1996) conducted an experimental and computational study for the heat transfer in rotor-stator system for a range of mass flow rate parameter $2530 \leq C_w \leq 9680$ at a fixed value of rotational Reynolds number $Re_\Omega = 1.25 \times 10^6$. The same geometrical configuration and experimental data has been used in the present simulation.

In this work, particular emphasis has been given to the simulation of the fluid flow and heat transfer in the rotor-stator disc system as shown in Fig. 1, where two-disc flow forms a model problem for the rotor-stator disc flows system. Two models, the low Reynolds number second moment closure and the low Reynolds number k- ϵ model have been applied for the fluid flow and heat transfer simulations in the rotor-stator disc system. These turbulence models were built-in by default in the Fluent commercial code. (Javiya *et al.* 2012) used different CFD commercial codes, including the Fluent for analysis of the flow and heat transfer in a direct transfer preswirl system and found a good agreement with the measurements. (Roy *et al.* 2001) is also used the Fluent software for the simulation of the convection heat transfer in a model rotor-stator disk cavity. The simulated results of convective heat transfer coefficient in good agreement with measurements.

++ corresponding authors: A. F. ABBASI

*Quaid-e-Awam University of Engineering, Science and Technology, Nawabshah, Pakistan

2. Problem Specification and Boundary conditions

The geometric dimensions considered here are the same as those used in the relevant experimental work of (Chen *et. al.* 1996) at the gap ratio $G=0.12$, rotational Reynolds number of order $Re_{\omega}=1.25 \times 10^6$ and for the range of mass flow rates $2530 \leq C_w \leq 9680$. The geometric dimensions are $b=0.391\text{m}$ $s=0.047\text{m}$ $r_i=0.0376\text{m}$ $S_c=0.004\text{m}$ No-slip boundary conditions are assumed on the solid walls. The cooling air at temperature $T_1=30^\circ\text{C}$ is assumed to enter the cavity axially from the centre of the stator and leaves radially through an axial gap between the shrouds with uniform inlet (U_i) and outlet (V_o) velocities. The radial distribution of the temperature on the rotor disc assumed to be non-uniform and increased with the radius of the disc and reached a maximum value of $T_s=100^\circ\text{C}$ near the periphery of the system, while the stator and shrouds were supposed to be adiabatic. The constant turbulent Prandtl number (i.e. $\sigma_h=0.9$) approach has been adopted for the heat transfer predictions.

$$\begin{aligned} U=V=W=0 & \quad \text{at } z=0, r_i < r < b \\ V=W=0, U=U_i & \quad \text{at } z=0, 0 < r < r_i \\ U=V=0, W=\Omega r & \quad \text{at } z=s, 0 < r < b \\ U=W=0, V=V_o & \quad \text{at } r=b, 0 < z < S_c \end{aligned}$$

3. NUMERICAL METHOD

The computational procedure employs a finite volume technique and adopts the line-by-line method based on the tri-diagonal matrix algorithm (TDMA) to solve the discretised transport equations. This numerical method also adopted the SIMPLEC algorithm of (Doormal and Raithby 1984) to update the pressure field and correct the velocity values to satisfy the mass continuity. For detail see references, (Memon

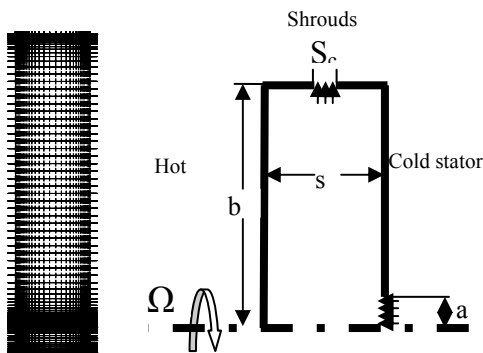


Fig 1. Schematic diagram for rotor stator system

Fig 2. Non-uniform grid arrangement

The near-wall (discs and shroud) refinement has been set to provide the non-dimensional wall distance (i.e. $y^+=yU\tau/\nu$, where y , $U\tau$ and ν are generalised the wall distance, friction velocity and kinematic viscosity

respectively) between the respective wall and the first nearest node as $y^+ < 0.5$, which ensures a fine distribution of the nodes in the near-wall spaces. A more accurate and economical solution can be obtained with a practice of non-uniform grids of 75×92 nodes in the axial and radial directions respectively, as shown in (Fig. 2).

4. Governing equations

Following are the governing equations of two turbulence models, the low Reynolds number $k-\epsilon$ model and the low Reynolds number second moment closure along with the energy equation, which are built-in by default in the Fluent commercial code. The details of the turbulence modelling and the energy equation are described hereunder:

4.1 Common form of governing equation

$$\frac{\partial}{\partial z}(\rho U \Phi) + \frac{1}{r} \frac{\partial}{\partial r}(r \rho V \Phi) = \frac{\partial}{\partial z} \left(\Gamma_{\Phi, z} \frac{\partial \Phi}{\partial z} \right) + \frac{1}{r} \frac{\partial}{\partial r} \left(r \Gamma_{\Phi, r} \frac{\partial \Phi}{\partial r} \right) + S_{\Phi} \quad (1)$$

with the continuity equation

$$\frac{\partial}{\partial z}(\rho U) + \frac{1}{r} \frac{\partial}{\partial r}(r \rho V) = 0$$

4.2 Low Reynolds number $k-\epsilon$ model

A well known two equation low Reynolds number $k-\epsilon$ model of (Morse 1991) (based on the transport equation for the kinetic energy (k) and its dissipation rate (ϵ))

$$\rho \overline{u_i u_j} = 2/3 \delta_{ij} \rho k - \mu_T \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} - 2/3 \delta_{ij} \nabla \cdot \underline{V} \right) \quad (2)$$

where $\nabla \cdot \underline{V}$ is the divergence of the velocity vector,

$$\nabla \cdot \underline{V} = \frac{\partial U}{\partial z} + \frac{\partial V}{\partial r} + \frac{V}{r} \quad (3)$$

$2/3 \delta_{ij}$ = Normal stresses (i.e. $i=j$ only)

4.3 Low Reynolds number second moment closure

The low Reynolds number second moment closure, which is based on the suggestions of Lai and So (1990), the so-called Reynolds stress closure, can be written in Cartesian tensor notation as

$$\frac{\partial}{\partial x_j}(\rho U_i U_j) = - \frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_i} \left(\mu \frac{\partial U_i}{\partial x_j} - \rho \overline{u_i u_j} \right) \quad (4)$$

where, U_i and u_i are the i th components of the mean and fluctuating velocity respectively, P is the mean static pressure, ρ and μ are the fluid density and viscosity respectively. The term $\overline{u_i u_j}$ represents the six components of the Reynolds stress tensor that can be determined by the turbulence model.

$$\frac{\partial}{\partial x}(\rho U \Phi) + \frac{1}{r} \frac{\partial}{\partial r}(r \rho V \Phi) = \frac{\partial}{\partial x} \left(\mu \frac{\partial \Phi}{\partial x} \right) + \frac{1}{r} \frac{\partial}{\partial r} \left(r \mu \frac{\partial \Phi}{\partial r} \right) + S_{\Phi} \quad (5)$$

4.4 Energy equation

The generalized form of the Reynolds-averaged energy equation for a steady turbulent flow can be written in cartesian tensor notation as,

$$\frac{\partial}{\partial x_j}(U_j \phi) = \frac{\partial}{\partial x_j} \left[\frac{\mu}{Pr} \frac{\partial \phi}{\partial x_j} - \overline{u_j \phi} \right] \quad (6)$$

The above governing transport equation for the conservation of momentum and energy for a axisymmetric and steady flow can be written in the cylindrical polar-coordinate system and illustrated in universal form for a general variable(ϕ) as,

$$\frac{\partial}{\partial x}(U\phi) + \frac{1}{r} \frac{\partial}{\partial r}(rV\phi) = \frac{\partial}{\partial x} \left(\frac{\mu}{Pr} \frac{\partial \phi}{\partial x} \right) + \frac{1}{r} \frac{\partial}{\partial r} \left(\frac{r\mu}{Pr} \frac{\partial \phi}{\partial r} \right) + s_\phi \quad (7)$$

Where the turbulent heat flux tensor emerged in equation (7) is represented by the source term (s_ϕ).

5 Discussion of Numerical Results

This section presents the simulated and experimental results for the heated rotor-stator disc system.

5.1 Computed flow structure

(Fig. 3). presents the comparison of computed streamlines of two models for the range mass flow rates, $2530 \leq C_w \leq 9680$ at fixed rotational Reynolds number, $Re_\square = 1.25 \times 10^6$. For a lower mass flow rate, $C_w = 2530$ a larger recirculation core of the fluid is formed in the cavity with the smaller source region; consequently a weaker wall jet is developed on the rotor disc surface, as shown in Fig3a. Whereas for a 35% higher mass flow rate, $C_w = 3900$, the size of the source region is increased result of that, comparatively a strong wall jet is formed on the rotor disc surface. This reveals the effects of mass flow rate on the computed flow structure, as shown in Fig3b. For a 59% higher mass flow rate, $C_w = 6100$ both the models predicted a strong wall jet on the rotor disc surface and two recirculation cores of fluid are formed in the inner and outer regions of the cavity. The size of the core rotation is contracted at the outer region of the cavity, which is smaller than that of previous cases of lower mass flow rate, as shown in figure 3c. This trend of change in flow structure further evident from a 74% higher mass flow rate, $C_w = 9680$. For this highest mass flow rate, the computed streamlines show the stronger wall effect on the rotor disc side, a weaker and contracted recirculation core is formed in the outer region of the cavity. However, both the models predicted the recirculation core of fluid in the source region, which attributed that for a higher mass flow rate, the incoming air impinged on the rotor disc with high intensity. A part of the air travels along the rotor disc in the shape of strong wall jet and remaining part of the air circulated in earlier stage of the cavity, as shown in figure 3d. A comparison of the two models illustrates the source-sink flow in the cavity and shows the same level of agreement in all the flow regions.

5.2 Computed static temperature contours

(Fig.). presents the comparison of the computed static temperature contours for the range of mass flow rate, $2530 \leq C_w \leq 9680$ at a fixed rotational Reynolds number

$Re_\square = 1.25 \times 10^6$. For a lower mass flow rate $C_w = 2530$ both the models predicted the larger recirculating core of fluid in the cavity with weaker wall jet and smaller source region. This weaker fluid jet not show the significant effect on the temperature distributions, consequently, minimum heat is transferred from the hot disc to incoming cooling air at lower part of the cavity and maximum heat is transferred in the outer part of the cavity, as shown in figure 4a (i) and (ii). For a 35% higher mass flow rate, $C_w = 3900$ the profound effects are seen on the temperature distribution between the hot disc and the cooling air at early stage of the cavity. Result of that more heat is transferred from the rotor(hot) disc to the air than to the cold stator, as shown in figure 4b (i and ii). Further 59% increased in mass flow rate $C_w = 6100$, effect on the flow structure, particularly on the wall-jet, which became more strong. This stronger wall jet effect on the temperature distribution from the hot disc to cooling air. Therefore, heat transfer rate is increased in the outer region of the cavity, as shown in (Fig. 4c) (i and ii). For the highest mass flow rate, $C_w = 9680$, the source region and the wall-jet are more dominant than the previous cases of lower mass flow rate. Therefore, higher temperature distribution is seen in the cavity space. The most significant effect of a higher mass flow rate is noted in the outer region of the cavity, where more heat is transferred from the hot disc to air than to the cold disc, as shown in (Fig. 4. d i and ii). The same effects of a strong wall-jet on heat transfer calculations were noted by (Pelle *et. al.* 2009). Overall, comparison of the computed static temperature contour shows that the radial distribution of the temperature depends on the mass flow rate, higher the mass flow rate ($C_w = 9680$), higher the temperature distribution and lower the mass flow rate ($C_w = 2530$) the lower the temperature distribution from the hot rotor disc to the air. This revealed that for higher mass flow rate the maximum temperature distribution take place in the cavity.

5.3 Computed Nusselt number

For the computation of the local Nusselt numbers, the inlet temperature is assumed adiabatic disc temperature as used in the definition of the measured Nusselt numbers. Therefore, the local Nusselt numbers may be expressed as:

$$Nu = \frac{r q_s}{k_f (T_s - T_f)} \quad (10)$$

(Fig. 5 a-d) show the comparisons between the computed and the measured local Nusselt numbers in radial direction for the range of mass flow rate, $2530 \leq C_w \leq 9680$ and for a fixed value of rotational Reynolds number, $Re_\square = 1.25 \times 10^6$ as adopted in the referred experimental work of (Chen *et. al.* 1996). For a lower mass flow rate $C_w = 2530$, the comparison of both the models shows a

good level of agreement with the measurement; however both the models are under estimated in the source region, as shown in figure 5a. This difference indicated that the flow is more complex in the inlet source region; therefore exact temperature values are not predicted by both turbulence models at this region. Another reason of this discrepancy may be the involvement of various uncertainties and assumptions in the experimental work. For a 34% higher flow rate, $C_w = 3900$, both the models again underestimated in the source region and there is about 10% increased in the local Nusselt numbers in the outer region of the cavity, as shown in figure 5b. This trend of increased in the local Nusselt numbers attributed to the higher mass flow rate, the extended size of the source region with the enhanced wall-jet strength on the rotor disc surface, consequently, the heat is transferred from the hot disc (rotor) to the wall-jet and then to the core region. Both models show a good agreement with the measurement; however the k- ϵ model is over predicted in the outer region of the cavity. For a 59% higher flow rate, $C_w = 6100$, the comparison of the computational and experimental results show the 24% increased in the local Nusselt numbers in the outer region of the wheel-space, as shown in (Fig. 5c). This enhancement in the local Nusselt numbers signifies the significant effects of higher mass flow rate in terms of a strong wall-jet. This attributed that the strong wall-jet carries the huge amount of heat and increase the heat transfer from the hot disc surface. The comparison shows that both the models are underestimated in the source region and overestimated in the outer region of the cavity. For a 74% highest mass flow rate, $C_w = 9680$, both the models show the same trend of variation in the local Nusselt numbers, which increased to 33% in the outer region of the cavity, as shown in figure 5d. For this highest mass flow rate, (Chen *et. al.* 1996) indicated that the distribution of the local Nusselt numbers on the hot disc was similar to that for the free-disc case. Comparison of two models shows the same discrepancies as noted in previous cases, where both the models underpredicted in the source region and over predicted in the outer region of the cavity. Overall, comparison of the two models for the range of mass flow rate revealed that the mass flow rate have the significant effects on the rate of heat transfer from the hot disc to cold disc which is increasing monotonically.

5.4 Computed Moment Coefficients

The performance of the two turbulence models is assessed through the comparison of computed moment coefficients $C_m = \frac{M}{1/2 \rho \Omega^2 b^5}$

where M is the frictional torque on the rotor surface) for a fixed value of rotational Reynolds number $Re_\Omega = 1.25 \times 10^6$ and the range of mass flow rates, $2530 \leq C_w \leq 9680$ as shown in table 1. Due to unavailability of the moment coefficient data, both the

models are compared through percentage of variation in the computed results. (Gan *et. al.* 1994 Daily and Nece 1960 and Kilic *et. al.* 2003) also reported the effects of mass flow rate on the moment coefficient on the rotor disc. This fact is also obvious here for the range of mass flow rate, which shows that for the higher mass flow rate, the value of the computed moment coefficient is higher than that of a low mass flow rate. (Memon 1995) revealed that the effect of superposed flow is to reduce the core rotation and the radial inflow on the stator, which consequently increase the moment coefficients on the rotor disc. This fact is well supported here by the higher predicted moment coefficients on the rotor disc surface. A comparison of the two models shows that the value of computed moment coefficients increases monotonically as mass flow rate increases. For all the values of mass flow rates, the low Reynolds number second moment closure overall predicts a 12% higher value of moment coefficient than that of the k- ϵ model, as presented in (Fig 6 and Table 1). This variation in the computed moment coefficient revealed that due to the enhanced wall treatment, the low Reynolds number second moment closure mimic the correct behaviour of the flow and captured the steep velocity gradient close to the rotor disc surface.

6.

CONCLUSIONS

The simulation of the symmetrical (non-isothermal) heated rotor-stator disc flow system were conducted for increasing temperature profile at a fixed value of rotational Reynolds number, $Re_\Omega = 1.25 \times 10^6$ and the range of mass flow rate, $2530 \leq C_w \leq 9680$. To assess the performance of the two models for these flow conditions, the computed streamlines of the two models illustrated the basic flow structure of four regions: source, core, boundary layers and sink as mentioned in the previous work of (Memon 1995), (Jacques *et. al.* 2002), (Poncet *et. al.* 2009) (Launder *et. al.* 2010). The computed streamlines of both the models illustrated a large recirculating core of fluid in the cavity which was shrinking at higher mass flow rate and formed a strong wall-jet on the rotor disc surface. Consequently, the higher temperature distributions were seen in this region, which results higher heat transfer from the hot disc to the air than to the cold stator disc. The level of the local Nusselt numbers also increased in the cavity on the higher mass rate. The comparison of the computed moment coefficients of two models, illustrated the variation in the computed moment coefficient with the value of mass flow rate. This variation is in accordance with the theoretical concepts, which stated that the computed moment coefficient varies according to two parameters, the mass flow rate and rotational Reynolds number. These parametric effects on the moment coefficients are well reported in the previous work of (Morse, 1991 and Memon 1995).

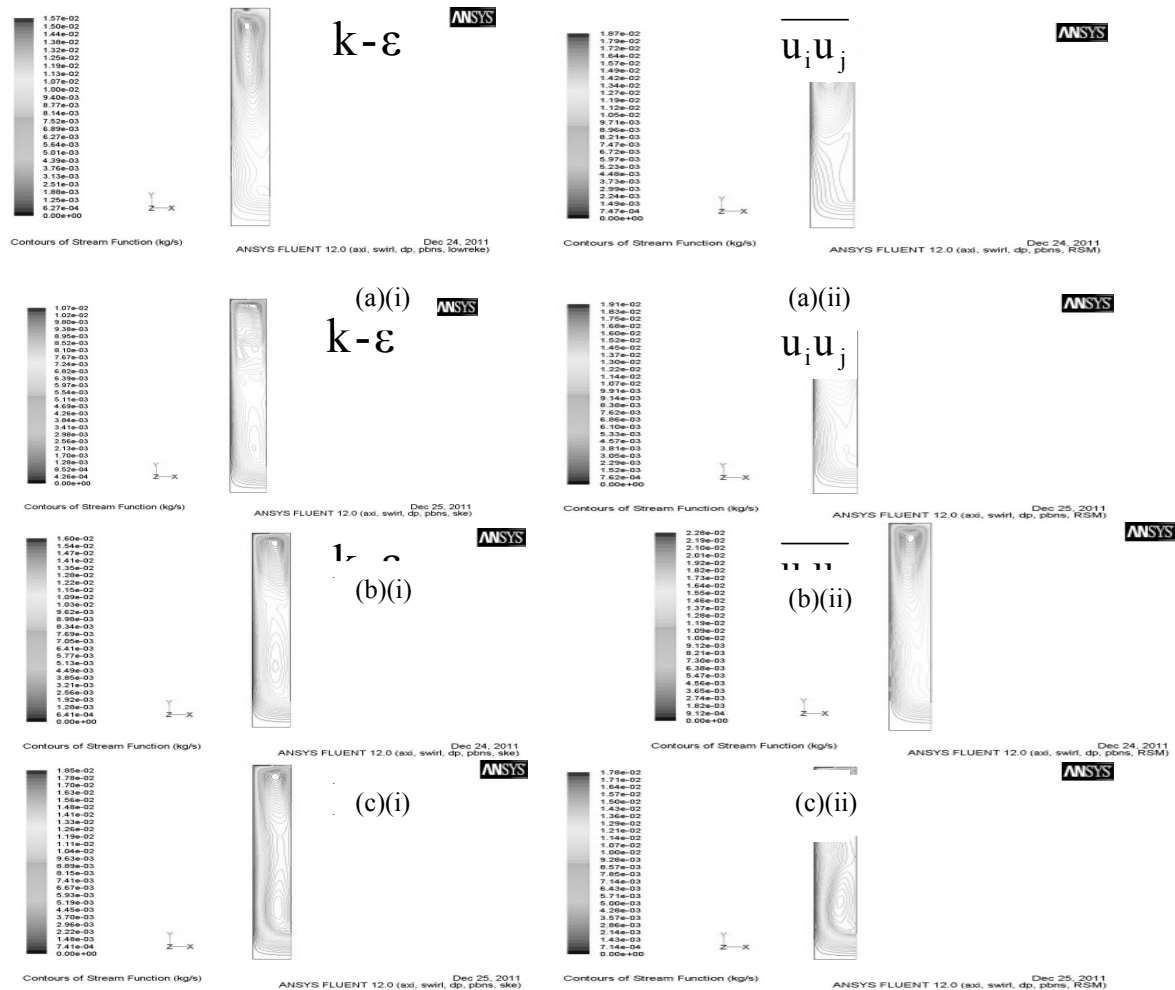
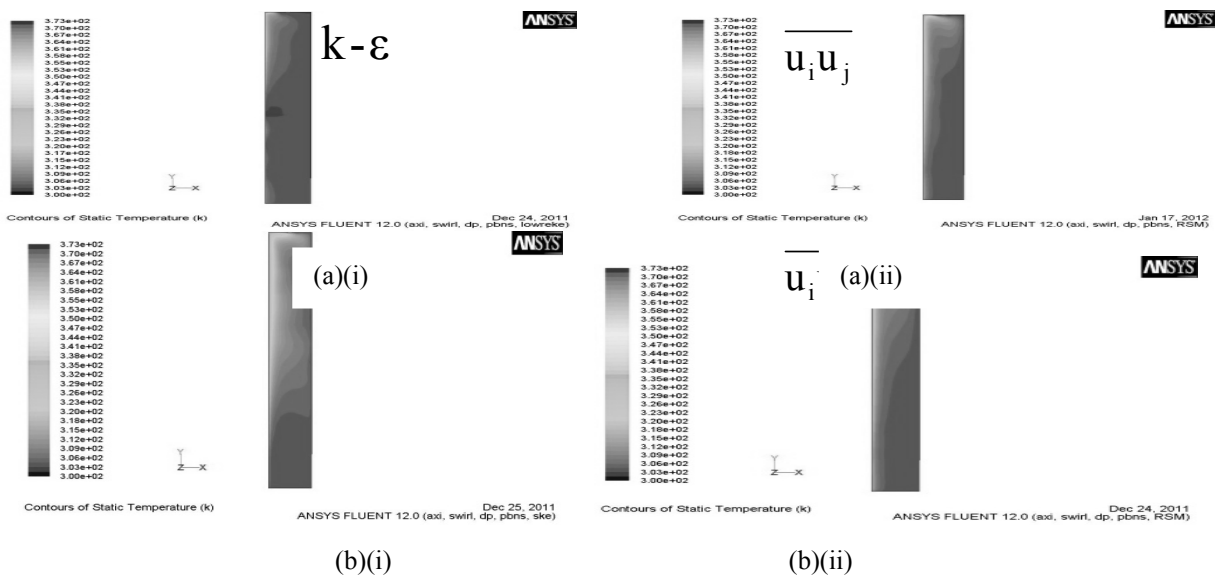


Fig. 3 Effect of mass flow rate parameter on computed streamlines for rotor-stator flow. The figure is organized into four rows (a, b, c, d) and two columns (i, ii). Each row corresponds to a specific mass flow rate parameter: (a) 1.0×10^6 (i) Low Reynolds number $k-\epsilon$ (d)(i) Low Reynolds number second moment $u_i u_j$ (d)(ii) Fixed rotational Reynolds number $Re_\Omega = 1.25$ $Cw=2530$ (b) $Cw=3900$ (c) $Cw=6100$ (d) $Cw=9680$



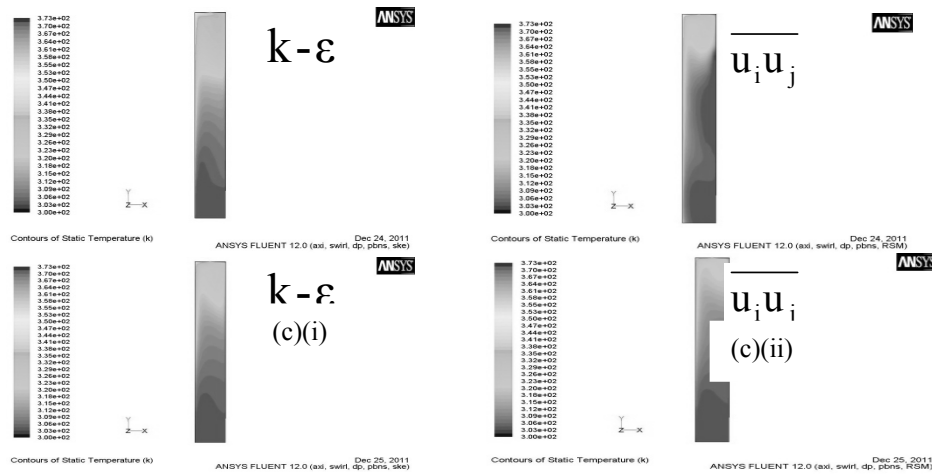


Fig. 4 Effect of mass flow rate parameter on the computed static temperature for rotor-stator system at fixed rotational Reynolds number $Re_{\theta} = 1.25 \times 10^6$ (i) Low Reynolds number $k-\epsilon$ model (ii) Low Reynolds number second moment closure (a) $C_w = 2530$ (b) $C_w = 3900$ (c) $C_w = 6100$ (d) $C_w = 9680$

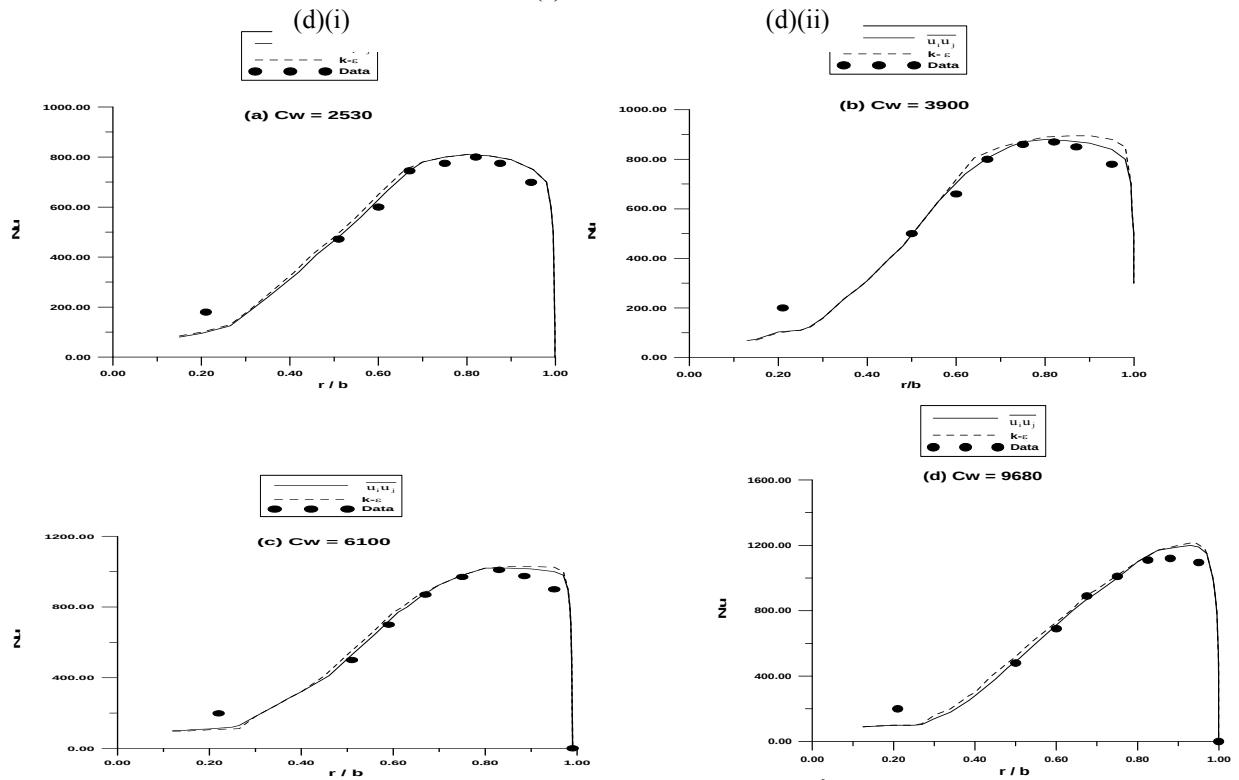


Fig. 5. Computed and measured Nusselt number for $Re_{\theta} = 1.25 \times 10^6$ and $G = 0.12$

Table 1 Comparison of predicted moment coefficients

Re_{θ}	C_w	$\overline{u_i u_j}$ ($C_m \times 10^3$)	$k-\epsilon$ ($C_m \times 10^3$)
1.25×10^6	2530	3.488	3.061
	3900	3.881	3.409
	6100	4.353	3.799
	9680	4.832	4.248

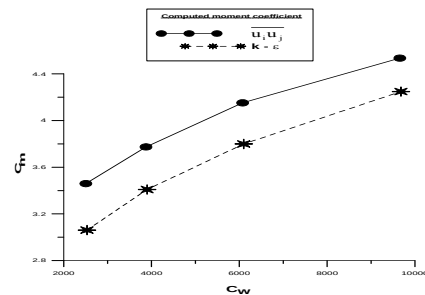


Fig. 6 Comparison of computed moment coefficient for the range of mass flow rates $2530 \leq C_w \leq 9680$

NOMENCLATURE

b	outer radius of cavity
$C_m = \frac{M}{1/2 \rho \Omega^2 b^5}$	moment coefficient
$C_w = \frac{m}{4\pi b}$	non-dimensional mass flow rate
$G = s/b$	cavity gap ratio
k	turbulent kinetic energy
M	frictional moment
Nu	Nusselt number
p	static pressure
q	heat flux
$Re_\theta = \Omega b^2 / \nu$	rotational Reynolds number
Re_τ	turbulent Reynolds number
r	radial co-ordinate
r/b	non-dimensional radial co-ordinate
r_i	inner disc radius
S	source term
s	axial gap between the discs
S_c	clearance between shroud
T_i	inlet cooling air temperature
T_s	disc surface temperature
U_i	uniform inlet velocity
$U_\tau = \sqrt{\tau_w / \rho}$	friction velocity
V_o	uniform outlet velocity
U, V, W	mean velocity components in z, r, θ
$u_i u_j$	Reynolds stress tensor
y	generalised wall distance
$y^+ = y U_\tau / \nu$	wall-distance Reynolds number
u, v, w	fluctuating velocity components
z	axial co-ordinate
z/s	non-dimensional axial co-ordinate
ε	dissipation rate of turbulent kinetic energy
ν	kinematic viscosity
μ	dynamic viscosity
μ_τ	turbulent viscosity
ϕ	generalised transport variable
Ω	rotational speed of the disc
ρ	density
δ_{ij}	Kronecker delta
Γ	diffusivity
$\sigma_h =$	turbulent Prandtl number
α_T	eddy diffusivity of heat

7. ACKNOWLEDGEMENTS

The authors would like to acknowledge this work to Higher Education Commission, Ministry of Science and Technology Government of Pakistan. for providing an opportunity and financial support to fulfil this work.

REFERENCES:

Abbasi, A.F., M.D. Memon, and A. Baloch, (2012) "Modelling and Predictions of Isothermal Flow Inside the Closed Rotor-Stator Flows", Mehran University Research Journal of Engineering Technology, Vol. (31): 1, 83-94.

Chen, J.X., X. Gan and J.M. Owen (1996) "Heat transfer in an air-cooled rotor-stator System", J. Of turbomachinery, Vol. (118): 444-451.

Daily, J.W. (1960) "Chamber dimension effects on induced flow and frictional resistance of enclosed rotating discs", J. Basic Eng., Vol. (82): 217-231.

Gan, X., M. Kilic and J.M. Owen (1993) "Flow and heat transfer between gas turbine discs", AGARD-CP-527, 25.1-25.11.

Gan, X., M. Kilic, and J.M. Owen, (1994) "Superposed flow between two discs contra-rotating at differential speeds", Int. J. Heat Fluid Flow, Vol. (15): 438-446.

Jacques, R., P. Quere, O. Daube (2000) "Axisymmetric numerical simulations of turbulent flow in rotor-stator enclosures". Int. J. Heat, Fluid Flow Vol. (23): 381-397.

Javiya, U., J.W. Chew, N.J. Hills, L. Zhou, M. Wilson and G.D. Lock, (2012) "CFD analysis of flow and heat transfer in a direct transfer preswirl system", J. of turbomachinery, Vol. (134): 031017Pp.

Kilic, M. and J.M. Owen (2003) "Computation of flow between two discs rotating at different speeds". J. of turbomachinery, Vol. (125): 394-400.

Lai, Y.G. (1990) "On near-wall turbulent flow modelling", J. Fluid Mech., Vol. (221): 641-673.

Launder, B.E., S. Poncet, and E. Serre, (2010) "Transition and Turbulence in Rotor Stator Flows", Annual Review Fluid Mechanics, Vol. (42): 229-248.

Memon, M.D. (1995) "Numerical modelling and prediction of fluid flow and heat transfer in rotating disc geometries", D.Phil. Thesis, Univ. of Sussex.

Morse, A.P., (1991) "Application of a low Reynolds number $k-\varepsilon$ turbulence model to high-speed rotating cavity flows". J. Turbomachinery, vol. (113): 98-105.

Owen J.M. and R.H. Rogers (1995) "Rotating cavities, Flow and Heat Transfer in Rotating Disc Systems", V. 2: Rotating cavities, Research Studies Press/John Wiley Inc., New York.

Pelle, J. and S. (2009) Harmand, "Heat transfer study in a rotor-stator system air-gap with an axial inflow", Applied Thermal Engineering, Vol. (29): 1532-1543.

Poncet, S. and E. Serre (2009) "High-order LES of turbulent heat transfer in a rotor-stator cavity", Int. J. of heat and fluid flow, Vol. (30): 590- 601.

Roy, R. P. and J. Feng, (2001) "A Study of Convective Heat Transfer in a Model Rotor-Stator Disk Cavity", J. Turbomachinery, Vol. (123): 621-632, 2001.

Van D. J.P. and G.D. Raithby (1984) "Enhancement of the SIMPLE method for predicting incompressible fluid flow". Num. Heat Transfer, Vol. (7): 147-163.