RESEARCH ARTICLE   OPEN ACCESS

Flow Analysis in Two Pass Smooth Square Channel

R.T.SARATH BABU*, D.KRISHAIAH**
*(Department of Mechanical Engineering, SIETK College, Puttur, Chittoor Dist, AndhraPradesh, INDIA)
** (Department of Mechanical Engineering, SIETK College, Puttur, Chittoor Dist, Andhra Pradesh, INDIA)

ABSTRACT
This project aim is to presenting the air flow analysis in two pass smooth square channel experiments. The experiments were carried out on a horizontally oriented two pass smooth square channel with turn in sharp 180^0. The smooth square channel size was 2 cm. Calculations were performed by using CFD software "gambit &fluent". The predictions results of mass flow rate & heat transfer rate of the air flow inside the sharp 180° turn in square channel with using different Reynolds numbers (10000, 20000, 40000 & 60000) were compared against with experiment results obtained from based on journal. Computational fluid dynamics simulations were performed using **Gambit2.3.16&Fluent6.3.26** software with 2D, unsteady and turbulent flow conditions assumed. Three different turbulence models were used namely the k-ε, $k-\omega$ and Reynolds stress models in conjunction with non-equilibrium wall functions.
Keywords: CFD, Gambit, Fluent, Reynolds numbers, turbulent flow

I.   INTRODUCTION

For better power output and increased thermal efficiency, the combustor temperature of gas turbine engines has continuously increased since the first designs in the 1940s. In the early 1960s, the combustor temperatures exceeded the melting temperature of simple turbine airfoils, requiring material improvements and internal and external cooling schemes to protect vane and blade airfoils from damage. Typical modern blade designs receive a significant fraction of their cooling from heat transfer on the internal passages that distribute coolant to film cooling holes.

Gas turbines play an important role in power industry, and are now widely used in aircraft propulsion, land based power generation, and other industrial applications. It is well recognized that one way to increase power output and thermodynamic efficiency of gas turbine engine is to increase turbine inlet temperature. However with the increase of the turbine inlet temperature, the heat load transfer to the blade is increased. The blade can only survive if effective cooling methods are used to remove the heat load from turbine blades. Fig. depicts the typical cooling technology for internal and external zone

A common way to cool the blade tip is to adopt internal cooling by designing serpentine (two pass, three pass,multipass) channels with a 180 degree turn inside the blade (as shown in fig) Taking advantages of impinging and turning effects, tip can be cooled to a certain extent. To augment forced convection, advanced method of enhancing the internal convective cooling for the tip region, are required.

The blade lifetime and reliability will then be improved.

Internal forced flow through the channels connected by 180 degree bend is frequently encountered in various turbo machinery and heat exchanger devices. One of the most notable features in such system is the effect of the turn-induced secondary flow on the transport phenomena around the bend.

Energy losses in sharp 180^0 turn are caused by the formation of a separation zone, which appears just after the corner. The stability of this separation zone is questionable, which makes the numerical prediction of the flow in sharp 180^0 turn difficult. Owing to the complexity of the flow structure inside the i80^0 turn, coupled with the limitations of experimental measurement and CFD techniques, a complete understanding of the turbulent air flow behavior in such systems is lacking. This is particularly the case with the two pass smooth square channel sharp 180° turn bends. However, there are a number of detailed experimental and numerical studies related to the single phase air flows with curved 180° turn bends.

The investigations were aimed on two pass smooth square channel in CFD solutions for traditional RANS approaches using $k-\varepsilon$ and $k-\omega$ turbulence models, compared the simulated results of different Reynolds numbers like 10000, 20000, 40000 & 60000 with the experimental results measured.
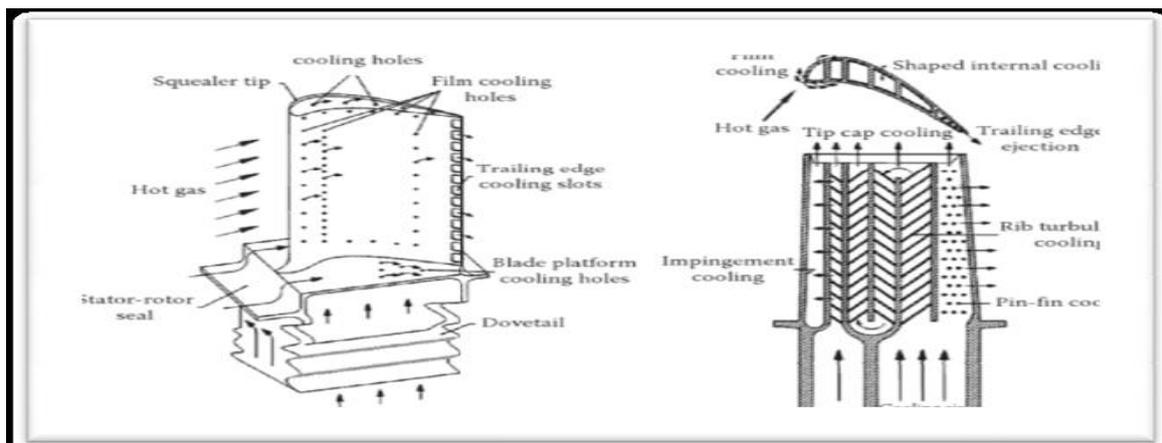
Fig.1.1 Typical cooling techniques for a turbine blade (a) internal cooling & (b) external cooling

II. CFD Theory

Computational Fluid Dynamics (CFD) is the science of predicting fluid flow, heat transfer, mass transfer, chemical reactions, and related phenomena by solving the mathematical equations which govern these processes using a numerical process (that is, on a computer).The result of CFD analyses is relevant engineering data used in:

➢ Conceptual studies of new designs
➢ Detailed product development
➢ Troubleshooting
➢ Redesign

From the 1960s onwards the aerospace industry has integrated CFD techniques into the design, R & D and manufacture of aircraft, engines. More recently the methods have been applied to the design of internal combustion engines, combustion chambers of gas turbines and furnaces. Furthermore, motor vehicle manufacturers now routinely predict drag forces, under-bonnet airflows and in the car environment with CFD. Increasingly, CFD is becoming a vital component in the design of industrial products and processes.

The ultimate aim of developments in the CFD field is to provide a capability comparable to other Computer Aided Engineering (CAE) tools such as stress analysis codes. The main reason why CFD has lagged behind is the tremendous complexity of the underlying behavior, which precludes a description of fluid flows that is at the same time economical and sufficiently complete. The availability of affordable high performance computing hardware and the introduction of user friendly interfaces have led to a recent upsurge of interest and CFD is poised to make an entry into the wider industrial community in the 1990s.

2.1CFD - How It Works

CFD applies numerical methods (called **discretization**) to develop approximations of the governing equations of fluid mechanics and the fluid region to be studied.

I.   Governing differential equations è algebraic
II. The collection of cells is called the **grid** or **mesh**.

2.2 HOW DOES A CFD CODE WORK?

CFD codes are structured around the numerical algorithms that can tackle fluid flow problems. In order to provide easy access to their solving power all commercial CFD packages include sophisticated user interfaces to input problem parameters and to examine the results. Hence all codes contain three main elements:

I.   A pre-processor
II. A solver and
III. A post-processor

2.2.1 Pre-processor

Pre-processing consists of the input of a flow problem to a CFD program by means of an operator-friendly interface and the subsequent transformation of this input into a form suitable for use by the solver. The user activities at the preprocessing stage involve:

➢ Definition of the geometry of the region of interest: the computational domain.
➢ Grid generation-the sub-division of the domain into a number of smaller, non- overlapping sub-domains: a grid (or mesh) of cells (or control volumes or elements).
➢ Selection of the physical and chemical phenomena that need to be modeled.
➢ Definition of fluid properties.
➢ Specification of appropriate boundary conditions at cells, which coincide with or touch the domain boundary.

2.2.2 Solver

There are three distinct streams of numerical solution techniques:

i.   Finite difference,
ii. Finite element,
iii. Spectral methods.

In outline, the numerical methods that form the basis of solver perform the following steps:

➤ Approximation of the unknown flow variables by means of simple functions.
➤ Discretisation by substitution of the approximations into the governing flow equations and subsequent mathematical manipulations.
➤ Solution of the algebraic equations.

2.2.3 Post-processor

As in pre-processing a huge amount of development work has recently taken place in the post-processing field. Owing to the interest popularity of engineering workstations, many of which have outstanding graphics capabilities, the leading CFD packages are now equipped with versatile data visualization tools. These include:

➤ Domain geometry and grid display,
➤ Vector plots,
➤ Line and shaded contour plots,
➤ 2D and 3D surface plots,
➤ Particle tracking,
➤ View manipulation,
➤ Color postscript output.

III. CFD Modeling of Turbulent Flows

A turbulent flow field is characterized by velocity fluctuations in all directions and has an infinite number of scales (degrees of freedom). Solving the NS equations for a turbulent flow is impossible because the equations are elliptic, non-linear, coupled (pressure-velocity, temperature-velocity). The flow is three dimensional, chaotic, diffusive, dissipative, and intermittent. The most important characteristic of a turbulent flow is the infinite number of scales so that a full numerical resolution of the flow requires the construction of a grid with a number of nodes that is proportional to $Re^{9/4}$

3.1 Modeling Turbulent Flows

Turbulent flows can be modeled in a variety of ways. With increasing levels of complexity they are:

o **Correlations**
 ❖ Moody chart, Nusselt number correlations
o **Integral equations**
 ❖ Derive ODE's from the equations of motion
o **Reynolds Averaged Navier Stokes or RANS equations**
 ❖ Average the equations of motion over time
 ❖ Requires closure
o **Large Eddy Simulation or LES**
 ❖ Solve Navier-Stokes equations for large scale motions of the flow. Model only the small scale motions
o **DNS**
 ❖ Navier-Stokes equations solved for all motions in the turbulent flow

3.1.1. RANS equations

➤ Velocity or a scalar quantity can be represented as the sum of the mean value and the fluctuation about the mean value as:

$$\phi = \overline{\phi} + \phi'(t)$$

➤ Using the above relationship for velocity(let f = u) in the Navier-Stokes equations gives (as momentum equation for incompressible flows with body forces).

$$\frac{\partial(\rho \overline{u}_i)}{\partial t} + \frac{\partial}{\partial x_j}\left(\rho \overline{u}_i \overline{u}_j + \rho \overline{u_i' u_j'}\right) = -\frac{\partial \overline{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial \overline{u}_i}{\partial x_j} + \frac{\partial \overline{u}_j}{\partial x_i}\right)\right]$$

➤ The Reynolds Stresses cannot be represented uniquely in terms of mean quantities and the above equation is not closed. Closure involves modeling the Reynolds Stresses.
 ➤ The RANS equations contain more unknowns than equations.
 ➤ The unknowns are the Reynolds Stress terms.
 ➤ Closure Models are:
o **Zero-equation** turbulence models
 ❖ Mixing length model
 ❖ no transport equation used
o **One-equation** turbulence models
 ❖ Transport equation modeled for turbulent kinetic energy k
o **Two-equation** models
 ❖ more complete by modeling transport equation for turbulent kinetic energy k and eddy dissipation e
o **Second-order** closure
 ❖ Reynolds Stress Model

3.1.2. k-☐ model

➤ The k-ε model focuses on the mechanisms that affect the turbulent kinetic energy (per unit mass) k.
➤ The instantaneous kinetic energy k(t) of a turbulent flow is the sum of mean kinetic energy K and turbulent kinetic energy k:

$$K = \tfrac{1}{2}\left(U^2 + V^2 + W^2\right)$$
$$k = \tfrac{1}{2}\left(\overline{u'^2} + \overline{v'^2} + \overline{w'^2}\right)$$
$$k(t) = K + k$$

➤ ε is the dissipation rate of k.
➤ If k and ε are known, we can model the turbulent viscosity as:

$$\nu_t \propto \vartheta\lambda \propto k^{1/2}\frac{k^{3/2}}{\varepsilon} = \frac{k^2}{\varepsilon}$$

➤ We now need equations for k and ε.

3.1.3. k-□ model
- This is another two equation model. In this model ω is an inverse time scale that is associated with the turbulence.
- This model solves two additional PDEs:
  - A modified version of the *k* equation used in the *k*-ε model.
  - A transport equation for ω.
- The turbulent viscosity is then calculated as follows:

$$\mu_t \;=\; \rho \frac{k}{\omega}$$

- Its numerical behavior is similar to that of the *k*-ε models.
- It suffers from some of the same drawbacks, such as the assumption that μ_t is isotropic.

3.1.4. Reynolds Stress Model
Reynolds Stress Transport Equation

$$\rho\, U_k\, \frac{\partial\, \overline{u_i u_j}}{\partial x_k} \;=\; P_{ij} \;+\; \Phi_{ij} \;-\; \varepsilon_{ij} \;+\; \frac{\partial J_{ijk}}{\partial x_k}$$

Generation:
(computed)

$$P_{ij} \equiv \overline{u_i u_k}\frac{\partial U_j}{\partial x_k} + \overline{u_j u_k}\frac{\partial U_i}{\partial x_k}$$

Pressure-Strain Redistribution:
(modeled)

$$\Phi_{ij} \equiv -\overline{p'\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)}$$

Dissipation:
(related to ε)

$$\varepsilon_{ij} \equiv 2\mu\,\overline{\frac{\partial u_i}{\partial x_k}\frac{\partial u_j}{\partial x_k}}$$

Turbulent Diffusion :

$$J_{ijk} = \overline{u_i u_j u_k} + \overline{p'\left(\delta_{jk}u_i + \delta_{ik}u_j\right)}$$

Equations written for steady, incompressible flow of body forces
- RSM closes the Reynolds-Averaged Navier-Stokes equations by solving additional transport equations for the Reynolds stresses.
  - Transport equations derived by Reynolds averaging the product of the momentum equations with a fluctuating property
  - Closure also requires one equation for turbulent dissipation
  - Isotropic eddy viscosity assumption is avoided

- Resulting equations contain terms that need to be modeled.
- RSM has high potential for accurately predicting complex flows.
  - Accounts for streamline curvature, swirl, rotation and high strain rates
    - Cyclone flows, swirling combustor flows
    - Rotating flow passages, secondary flows

IV. OVERVIEW OF FLUENT-GAMBIT
GAMBIT is a state-of-the-art preprocessor for engineering analysis. With advanced geometry and meshing tools in a powerful, flexible, tightly-integrated, and easy-to use interface, GAMBIT can dramatically reduce preprocessing times for many applications.

FLUENT is a computational fluid dynamics (CFD) software package to simulate fluid flow Problems. It uses the finite-volume method to solve the governing equations for a fluid. It Provides the capability to use different physical models such as incompressible or compressible, inviscid or viscous, laminar or turbulent, etc. Geometry and grid generation is done using GAMBIT which is the preprocessor bundled with FLUENT.

V. RESULTS AND DISCUSSIONS
5.1 Test section
The detailed measurement of turbulent flow carried out at a180 degree turn in a two pass square smooth channel, which was made from aluminum plate. The test section was a multipass channel with a 2cm square cross section. The top, the bottom, and the outer walls of the channel were constructed of 0.5 cm thick aluminum plates. The inner (divider) wall was constructed of two 0.5cm thick aluminum plates, bonded together back to back with double sided tape.

5.2 Experimental Apparatus and Instrumentation
The main components of the test apparatus are the test section, a settling chamber, a calibrated orifice flow meter, a control valve, and a centrifugal blower. The entire apparatus, together with the measuring instruments, was located in an air-conditioned laboratory, which was maintained at a constant temperature of 23°C throughout the tests.

Fig5.2. Experimental set up

5.2.2.Construction and assembly

Is as represented as 180^0 turn. The inlet and exit flow passage with aspect ratio of 1:1. In which the shorter wall of airfoil pressure and suction side. Each channel is 2X2 cm in size, with the hydraulic diameter of 2cm. Channel length to the channel width, X/D is 13.

A bell mouth inlet was used for the test model is used to reduce any inlet distortion. The inlet channel length is about thirteen hydraulic diameters. Both inlet and exit channel are smooth. Fig shows the assembly view of test model. First blower Supplies continuous amount of air at specific flow rate. This flow is controlled by use of control valve. After that the flow is measured by use of pressure differential instrument as orifice flow meter. The U-tube manometer is placed across the orifice for measurement of pressure difference. After that air flowing through the test section across the honey comb like straighter. Figure displaced to monitoring pressure [p] and temperature [Tc] is used in the model. The tip surface heat transfer coefficients are all based on the average temperature .which differ by only a few degrees due to the large flow rate and limited surface area. In the test model aluminum plate is placed over Nicrome thin foil plate type heater. For the measurement of average temp of the channel. Thermocouples are placed in aluminum test section.
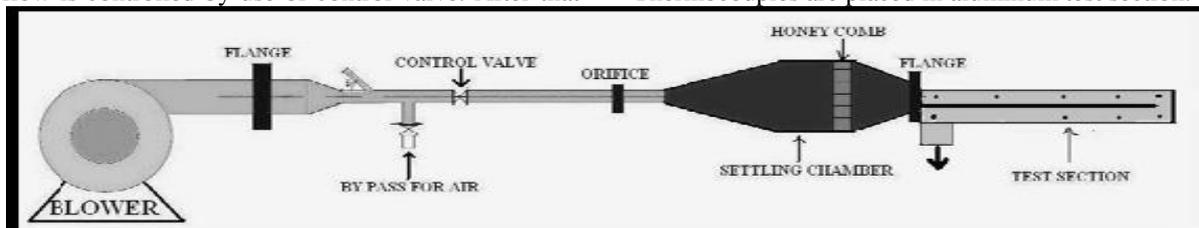
Fig5.3 Details of experiment setup

5.2.3 Experimentation

Experiment is conducted to find out total heat transfer and mass flow rate for smooth channel, dimensions of pattern 2mmX2mm (upstream and downstream). Experiment conducted for various Reynolds number ranges from 10000-60000. For the verification of experimental set up experiment conducted for smooth channel. To start the experiment we provide constant different Reynolds numbers. Measuring the pressure drop across the test section and orifice, we calculate the mass flow rate and total heat transfer rate for channel.

5.2.4 System Variables

Inlet temperature of Air: 23^oC
Reynolds number: 10000-60000
Prandtl number: 0.7
Hydraulic diameter: 2cm

5.2.5 Data Reduction

The aim of this experiment is to investigate the flow analysis for two pass smooth square channel used in turn of 180-degree bend. The procedure of normalizing the parameters is discussed in following section. The Reynolds number based on the channel hydraulic diameter is given by
$Re = v\ Dh\ /\ \nu$
$Re = \rho v Dh/\mu$
Where,
ρ density
v characteristic velocity (such as inflow)
L characteristic length (such as the length of an object in your flow)

μ viscosity

The average heat transfer coefficients are evaluated from the measured temperatures and heat inputs. With heat added uniformly to fluid (Qair) and the temperature difference of wall and fluid (Tw−Tb), average heat transfer coefficient will be evaluated from the experimental data via the following equations:
$Qair = Qconv = m\ Cp\ (To - Ti) = V\ I\ (2)$
$Qconv = h\ A\ (Tw-Tb)$
Thus
$h = Qconv\ /\ A\ (Tw-Tb)\ (3)$
in which
$Tb = (To + Ti)/\ 2\ (4)$
&
$Tw = \Sigma Ts/6$
Average Nusselt number is written as:
$$Nu = hD_h\ /\ K$$
In order to measure Nusselt number, the pressure difference between inlet and outlet must be obtained for smooth channel is given by Nu number as normalized by the corresponding values for fully developed turbulent flow heat transfer based on the inlet channel conditions. The fully developed channel Nu number was determined using the Dittus-Boelter correlation.
$$Nu = 0.023\ X\ Re^{\ 0.8}\ X\ Pr^{\ 0.4}$$
Mass flow rate of air in the two pass square channel is calculated theoretically. Net mass flow rate obtained by corresponding values for fully developed turbulent flow based on the inlet conditions. By using the following formula given below:
$$m = \rho Av$$
where,

m is mass flow rate in kg/s
ρ is density in kg/m^3
A is area of the cross section in m^2
V is velocity of air in m/s

5.3.Geometry Setup and Boundary Conditions

The two pass smooth square channel experiments, as shown in Fig. were constructed with GAMBIT Design Modeler, a pre-processor of the FLUENT code. For this project FLUENT 6.3.26 code has been selected and the main parameters of the materials are given in Table.1

Property	Aluminum(solid)	Air(fluid)
Density(kg/m3)	2719	1.225
Cp (Specific Heat)(j/kg-k)	871	1006.43
Thermal Conductivity(w/m-k)	202.399	0.0242
Viscosity(kg/m-s)	-	1.7894001e-05
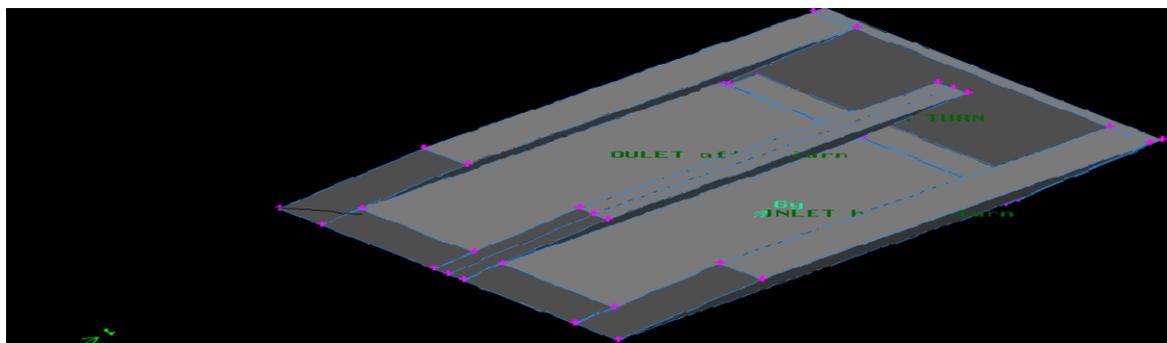
Fig 5.4 3D Design model in GAMBIT in Design module

Fig -5.5: 2D Design model in GAMBIT in Design module
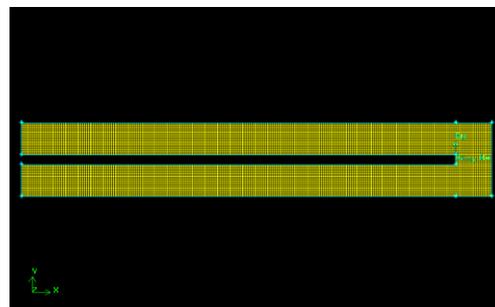
Fig-5.6: Meshing in GAMBIT

The unsteady-state 2d, pressure-based RANS simulation with the $k-\omega$, k-epsilon (k- ε) and Reynolds stress turbulence models are used in the simulation. Pr$_t$ is the turbulent Prandtl number arises from the application of the eddy viscosity hypotheses in the Reynolds averaging process of this transport equation. In this simulation used solver controls equations are Flow, Energy and Turbulence. Pressure velocity coupling is SIMPLE algorithm.

VI. Results
5.7.1 Different Reynolds number Comparison

Figure shows representative cross-sectional plots of the velocity, pressure distribution for the simulation by using $k-\omega$, k-ε and Reynolds stress turbulence models and the mass flow rate & heat transfer rate for the measurement. Predicted results

obtained by using the two pass smooth square channel of different Reynolds numbers (10000,20000,40000&60000) of air are generally in good agreement with the experimental data by using different Reynolds number. The following table shows the models and settings are used in these simulations.

Model Settings
Space 2D Time Unsteady, 1st-Order Implicit Viscous $k-\omega$, k-ε and Rsm model Wall Treatment Non-Equilibrium Wall Functions Heat Transfer Enabled
These flow is based on pressure and finite volume method is implicit , iterations taken 400 times, here below diagram are shows of the following pressure

and velocity distribution for Reynolds number s and for each compared with three turbulence models.

The study work included the comparison of various graphs for three turbulence models like $k-\omega$, k-ε and Reynolds stress conditions; the various graphs are plotted in the four Reynolds number ranges from 10000to 60000 .

Here, below shown figures are obtained from FLUENT CFD stimulation of post processing by using the three turbulence models compared of the **Case (i) Re=10000**

four Reynolds numbers like wise iterations, velocity profile contour with velocity distributions and pressure distributions.

In these analysis in the shows the flow separation in inner wall of channel at 180^0 sharp turn . for these reasons below figures are show inlet (before turn)of channel is greater flow rate than outlet(after turn) of channel .   here below results for Reynolds numbers(10000-60000) is each one camper to three turbulence models(K-ε, $k-\omega$, RSM model

K-ε model $k-\omega$ model   RSM model
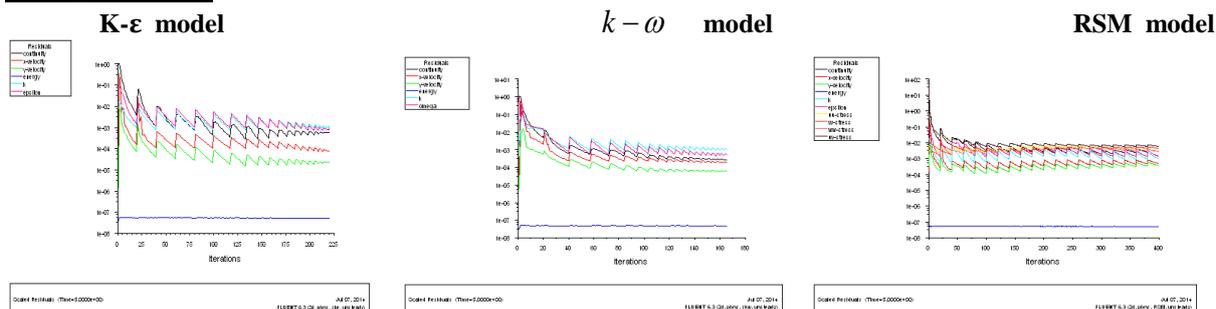
Fig-5.7:Iterations comparisons
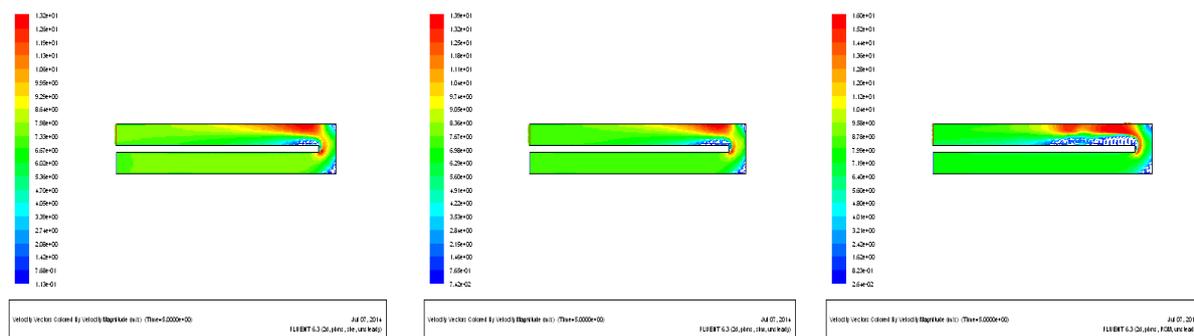
Fig-5.8: Distribution of velocity comparisons with three models
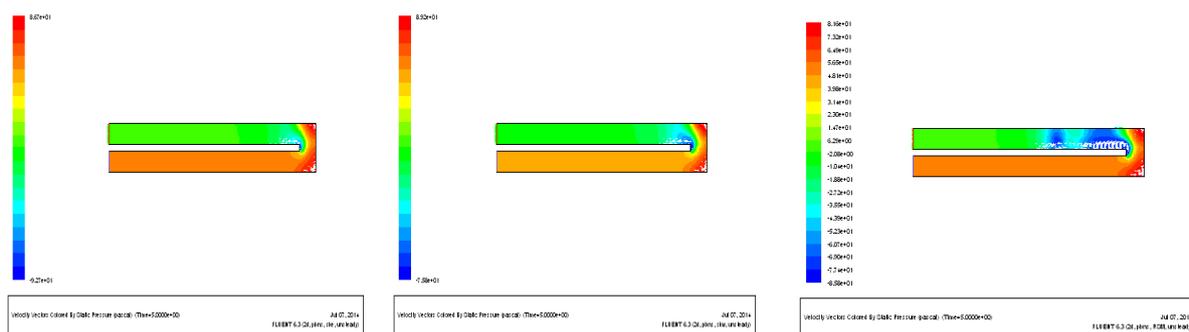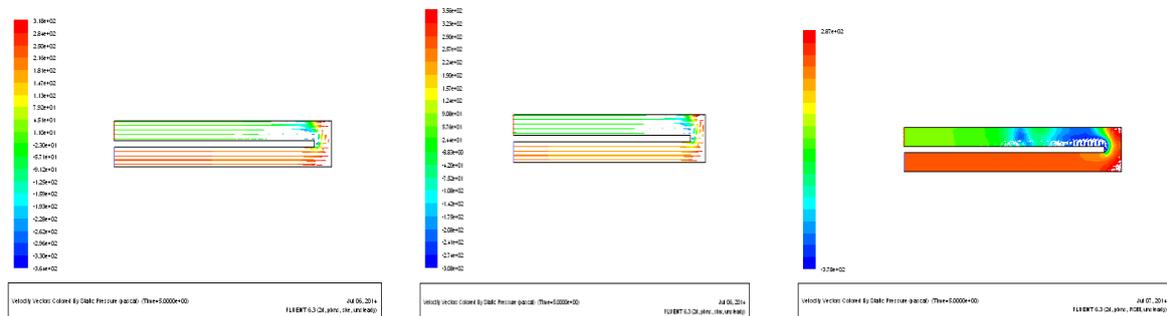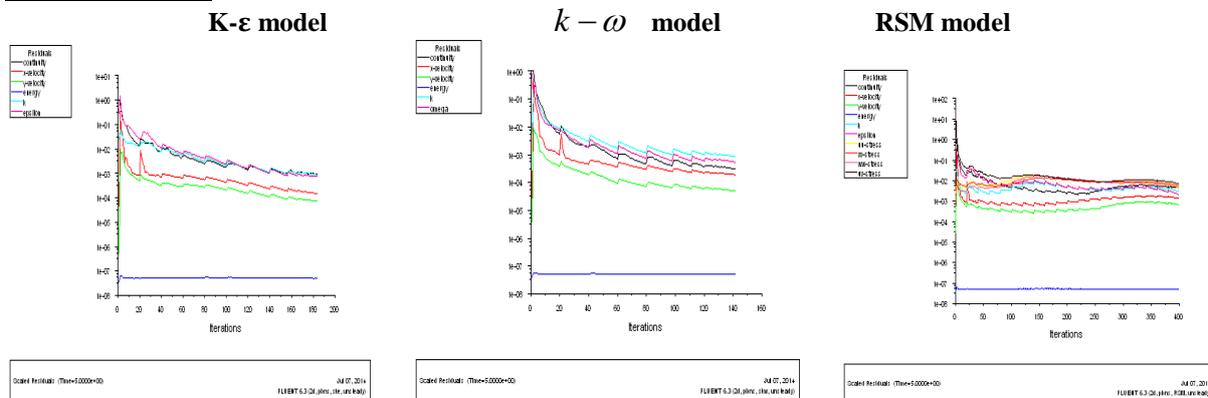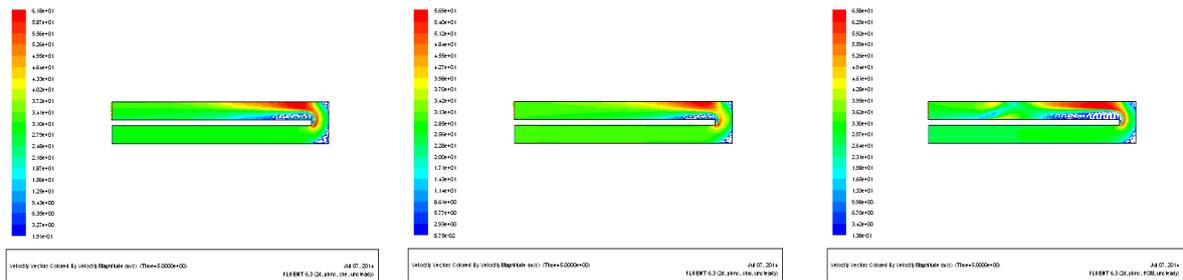

Fig-5.9: Distribution of pressure comparisons with three models

Case (2) Re=20000

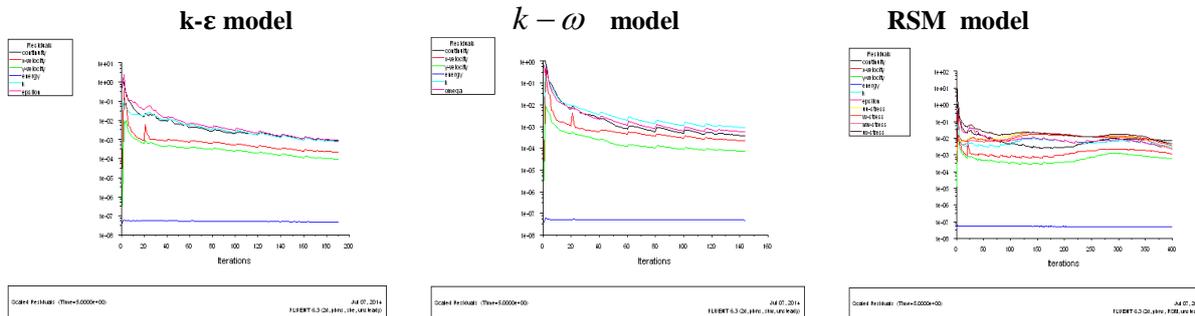| K-ε model	$k-\omega$ model	RSM model
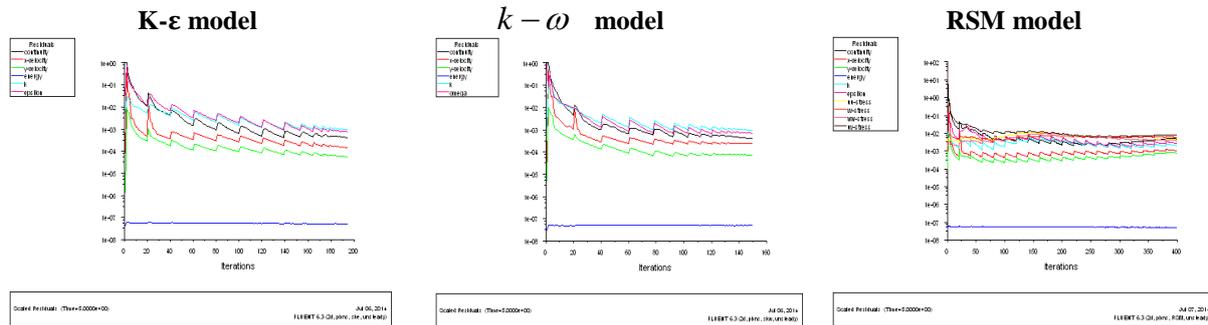
Fig-5.10:Iterations comparisons
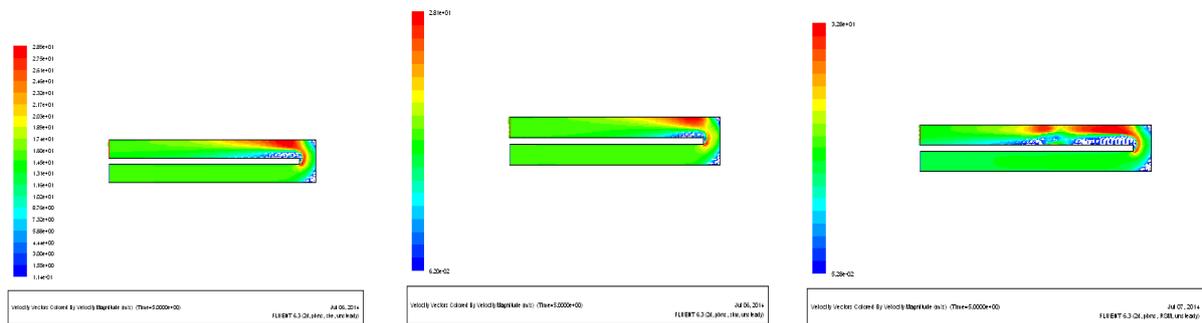
Fig-5.11:Distribution of velocity comparisons with three models

Fig-5.12: Distribution of pressure comparisons with three model

Case (3)Re=40000

K-ε model	$k-\omega$ model	RSM model

Fig-5.12:Iterations comparisons

Fig-5.12:Distribution of velocity comparisons with three models
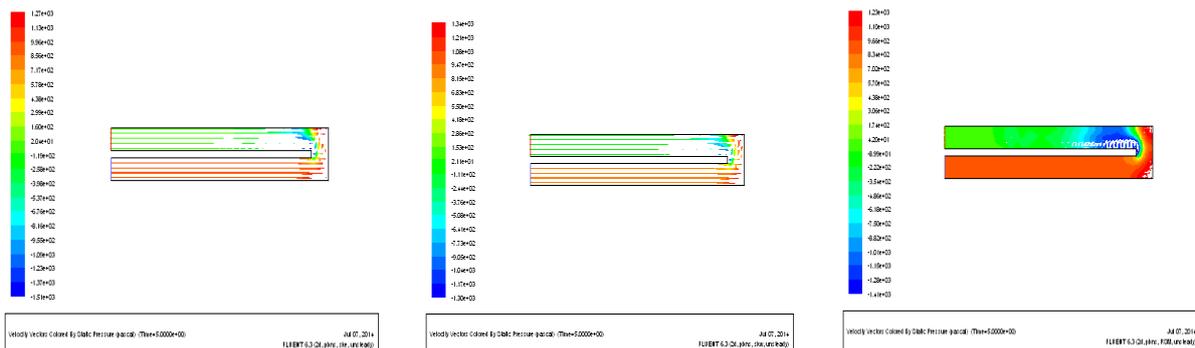
Fig-5.13:Distribution of pressure comparisons with three models

Case(4)Re=60000

| **k-ε model** | $k - \omega$ **model** | **RSM model** |

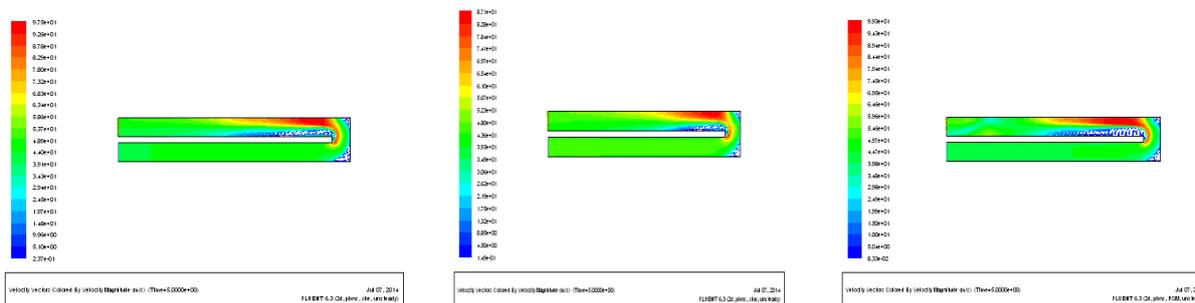Fig-5.14:Iterations comparisons

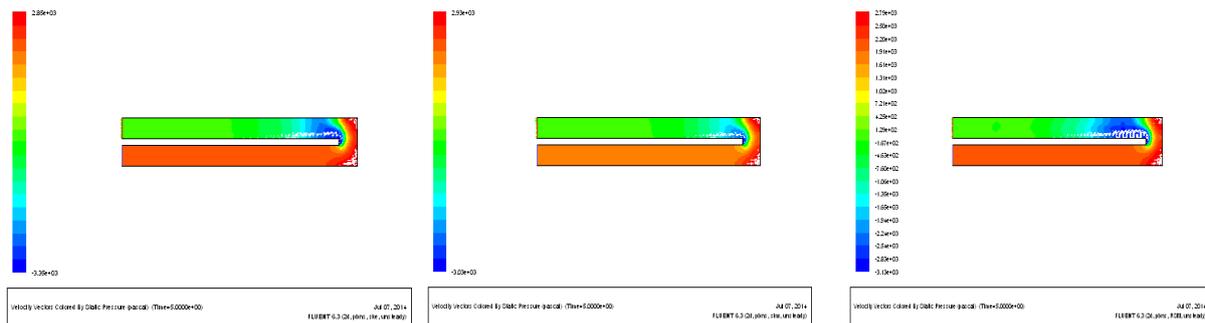**Fig-5.15: Distribution of velocity comparisons with three models**

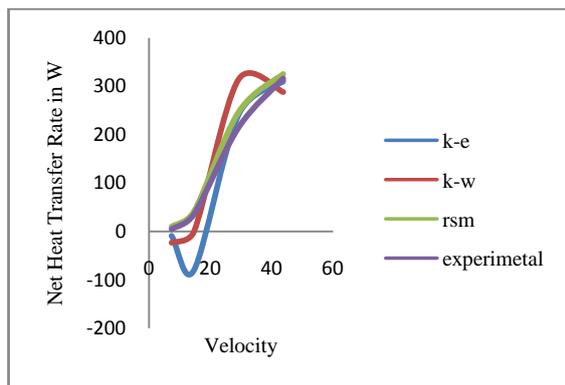**Fig-5.16: Distribution of pressure comparisons with three models**

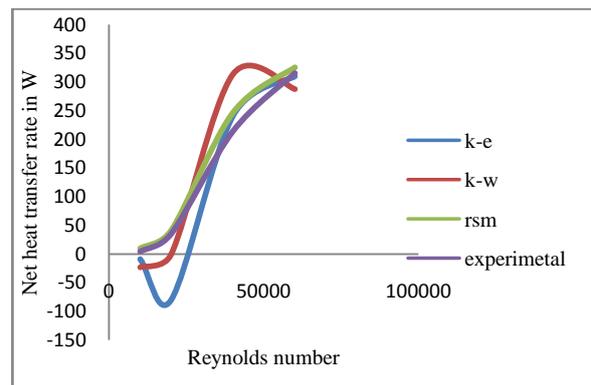Figure-5.17: Total heat transfer rate v/s velocity

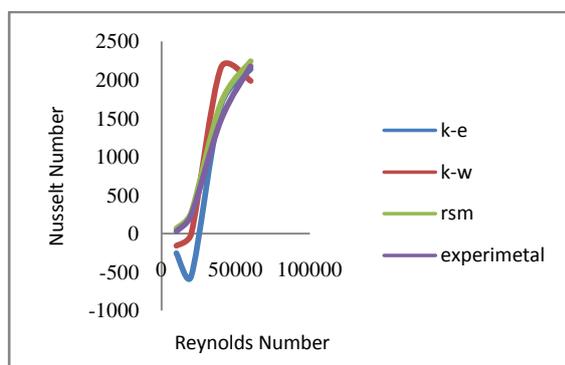Figure-5.18: Total heat transfer rate v/s Reynolds number

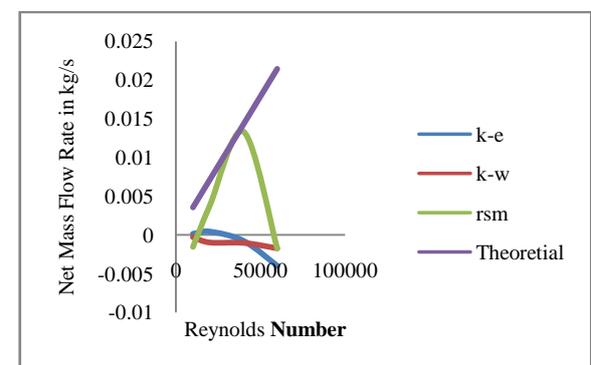Figure-5.19: Nusselt Number v/s Reynolds number

Figure-5.20: mass flow rate v/s Reynolds number

VII. Conclusions

Investigations have shown, that Reynolds averaging based RANS turbulence models like $k-\omega$ or $k-\varepsilon$ are able to satisfactorily predict the turbulent flow of air in two pass square smooth channel, while the flowing of air in sharp 180^0 turn of four different Reynolds number like 10000,20000,40000 & 60000 in two pass smooth square channel is a challenging test case for CFD methods, and advanced scale-resolving turbulence modeling approaches like LES (Large eddy simulation), DES (Detached eddy simulation) are required in order to simulate the flow and temperature fields.

The CFD results have been compared to the experimental data .

The numerical simulation of $k-\omega$,k-ε and Reynolds stress turbulence models for analyzing the flow in two pass smooth square channel. Some conclusions may be drawn as follows:

1. The simulation results are developed based on the $k-\omega$, k-ε and Reynolds stress turbulence models. Compared with the experimental results for heat transfer rate, are in slightly better results predicts from Reynolds stress models, experimental data.

2. Mass flow rate stimulation results obtained from based on the $k-\omega$, k-ε and Reynolds stress turbulence models is compared with theoretical results, are slightly better performance obtained from Reynolds stress model

7.1 Effect of Reynolds number in two pass smooth square channel

1. The effect of the flow separation (at the tip of the inner wall) and reattachment on the flow field can be seen very clearly in the inner-wall in the after-turn region. It shows the figure comparisons velocity, pressure distribution of three turbulence models of CFD.

2. the results figure are shows four Reynolds numbers effect of flow in pressure , velocity distribution with comparisons of three turbulence models

3. As per for above reference of the graphs depicts whenever increasing the Reynolds numbers the heat transfer rate and mass flow rates are increased.

7.2 Further future study of project

1. Study the local heat/mass transfer coefficients in a two-pass rib roughened channel.

2. Analysis of other advanced turbulence models direct numerical stimulations(DNS), large eddy stimulations(LES) for smooth and roughened channels

3. Study the effect of the channel aspect ratio1,2,3 on the local heat/mass transfer rate in two-pass ribbed channels.

4. Flow analysis in the two pass rib roughened channel for different ribs angle values like $30^0, 45^0, 60^0.$

5. Flow analysis in three pass ribbed channels

References
Journal Papers:

[1] *G.V.phadtare et al. / international journal of engineering science and technology (IJEST)"experimental investigations of heat transfer enhancement in tip of 180-degree bend of square duct with vortex generator"*

[2] *Local Heat/Mass Transfer and Pressure Drop in a Two-Pass Rib-Roughened Channel for Turbine Airfoil Cooling* by J.C. Han and P.R. Chandra Texas A & M University College Station, Texa*s*

[3] *an experimental and numerical convective heat transfer analysis over a transonic gas turbine rotor blade by keith baharath cassie KSME International Journal, Vol. 18 No. 7, pp. 1258~ 1266, 2004 ,*

[4] A Naphthalene *Sublimation Study on Heat/Mass Transfer for Flow over a Flat Plate by ,long-Hark Park ,Korea Atomic Energy Research Institute, Dukjin-dong, 150 Yusung-gu, Taejeon* 305-353, Korea

[5] *International Journal of Heat and Fluid Flow 31 (2010) 19–31"Heat transfer and pressure drop in two-pass rib-roughened square channels with bleed from sharp bend"* journal homepage: *www.elsevier.com/locate/ i jhf f*

[6] *comparison of continuous and truncated ribs on internal blade tip cooling by Proceedings* of *ASME Turbo Expo 2012*

Proceedings Papers:

[1] FLUENT_*tutorial_2008aug14fin*

[2] *David c. Wilcox ,"Turbulence modeling of CFD" ,1993-94 Edition,* page from 73 to 163, published by DCW industries Inc., La Canada, California