CFD Analysis of Flow through Turbine Blade in 2D Cooling Channel with Square Rib Turbulators

C T Dheeraj Kumar Singh, G Bharath Goud, J S P Naga Kumar

***Abstract*— In present day world the usage of gas turbine in power generation for industrial purpose has increased. This forced the researchers to concentrate more on the efficiency and the performance of the turbine. Most important feature includes the cooling of the turbine blades which are exposed to high temperature gas. If ignored there is every possibility that the turbine plates get damaged resulting in a heavy loss. In order to this, research has carried in several numbers to get the detail study with already existing methods like pin fin cooling, film cooling for leading edges. In this paper our efforts are mainly focused on the turbine blade internal cooling, which is the most crucial part where maximum heat exchange is possible. The cooling system should be for all possible parts of the turbine. The turbine blade tip is arranged with certain ribs based on p/e ratio. A comparative study was made in a square rib turbulators for Reynolds number 20000 and 30000 with p/e ratio as 10mm. A 2D model of the turbine blade internal is made using GAMBIT and simulation is processed in FLUENT and result of contours are made to study effectively. Heat transfer and flow measurement diameters calculated to show the effective change.**

***Index Terms*— pitch by eccentricity (p/e) ratio, Reynolds number, rib turbulators.**

I. INTRODUCTION

Turbine blade cooling is important to decrease the sharpened steel metal temperature to satisfactory levels for the materials expanding the warm ability of the motor. Because of the commitment and the advancement of turbine cooling systems, At a steady weight proportion, warm proficiency increments as the greatest temperature increments are considered. Be that as it may, high temperatures would harm the turbine, and the cutting edges are below huge radiating hassles and supplies are weak at great temperature. In this way, turbine edge refrigeration is fundamental. By enormously expanded the temperature capacity of turbine cutting edges. Further preparing routines like hot isostatic pressing enhanced the compounds utilized for turbine sharpened pieces of steels and expanded turbine cutting edge performance. Modern turbine blade sharp edges regularly utilize nickel-based super alloys that consolidate chromium, cobalt, and rhenium. Turbine blade edge refrigerating can be arranged in two good segments:

C T Dheeraj kumar singh, PG student, M. Tech in Computational Fluid Dynamics, University of petroleum and energy studies.

G Bharath Goud, PG student, Department of Mechanical Engineering, JNTUHCEH, Hyderabad.

J S P Naga kumar, B. Tech in Mechanical Engineering, St. Marys integrated campus, Hyderabad.

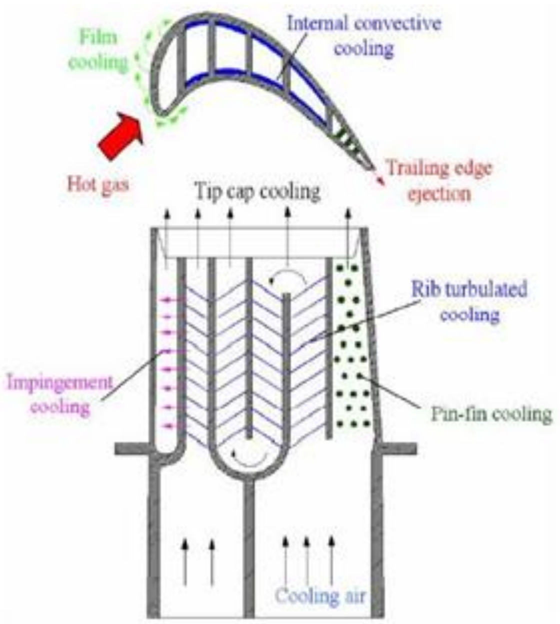

Figure: Cooling types of turbine blade

The internal, where the warmth(heat) is evacuated with a range of convection and impingement chilling arrangements, where large speed wind streams and works the internal surfaces of the turbine components and edges, and the external blade temperature, sharp edge with a specific end goal to make a dainty cooling layer. Here an extensive variation of inner plus outer cooling courses of act has been related previously; in any case is done.

II. COMPUTATIONAL DETAILS:

The comparisons overseeing smooth movement, the Navier-Stokes mathematical statements are shown. An outline of turbulence and heat exchange models with writing references are given. Extraordinary references to change because of velocity of flow affected turbulence are made. The mathematical statements that govern smooth movement of flow and heat exchange are the continuity, momentum and energy equations. Navier Stokes equations can be formed in either a conservative form, or in the non-conservative form structure. Discussing continuity, momentum and energy equations below.

Continuity equation:

$$\boxed{\frac{D\rho}{Dt} + \rho\nabla.\vec{V} = 0}$$

Momentum equation:
These momentum equations generally depend upon the Newton's second law:
F = m × a
F implies for force on the fluid particle. It again divides into two types those are
1) Pressure forces
2) Body forces

Pressure forces are stress forces i.e normal stress and shear stress on the particle and body forces is generally gravitational force. Here are momentum equations for x, y and z components these equations called as N-S equations:

X-momentum

$$\rho\frac{Du}{Dt}=-\frac{\partial p}{\partial x}+\frac{\partial\tau_{xx}}{\partial x}+\frac{\partial\tau_{yx}}{\partial y}+\frac{\partial\tau_{zx}}{\partial z}+\rho f_x$$

Y-momentum

$$\rho\frac{Dv}{Dt}=-\frac{\partial p}{\partial y}+\frac{\partial\tau_{xy}}{\partial x}+\frac{\partial\tau_{yy}}{\partial y}+\frac{\partial\tau_{zy}}{\partial z}+\rho f_y$$

Z-momentum

$$\rho\frac{Dw}{Dt}=-\frac{\partial p}{\partial y}+\frac{\partial\tau_{xz}}{\partial x}+\frac{\partial\tau_{yz}}{\partial y}+\frac{\partial\tau_{zz}}{\partial z}+\rho f_z$$

Energy equation:
The first law of thermodynamics expresses that the exchange of energy for a framework is the result of connected work and heat exchange through that region. In its most finish plan the energy comparisons is given

Here T_{ij} are the surface forces familiar to the pressure and viscous term from the momentum equations. The total internal energy is denoted with E_0 which includes kinetic energy.

III. GEOMETRY AND MESH DETAILS:

In this, the considered geometry two dimensional model. The hydraulic diameter D is considered as 25mm. In this case the

$$\frac{\partial\rho E_0}{\partial t}+\frac{\partial\rho U_i E_0}{\partial x_i}=\rho U_i F_i-\frac{\partial q_i}{\partial x_i}+\frac{\partial}{\partial x_j}(U_i T_{ij})$$

rib dimensions are considered with the relation of
e= 0.1 X D = 0.1 X 25 = 2.5mm and the
pitch(P) value is 25mm.
P/e ratio is calculated i.e = 25/2.5 = 10mm.
Here for this geometry the p/e ratio is considered to be 10mm. This means the distance from first rib begin to second rib start. This is maintained same in case of all the ribs present in the geometry. The inlet of the geometry is elongated because to occur fully developed flow till it reaches the rib turbulators. The outlet of geometry is maintained less when compared with inlet length of geometry. Orthogonal ribs i.e ribs perpendicular to the flow (90 Degrees to flow) and these ribs are parallel to each other. In the considered case 4 no. of ribs are placed in this geometry. Model of geometry is made GAMBIT. For this 2d geometry a better work has been done of finer mesh levels utilizing GAMBIT. Quad element is used and map mesh is done. Names of boundaries are also been mentioned like inlet as velocity inlet, outlet as pressure outlet and remained all done with wall.

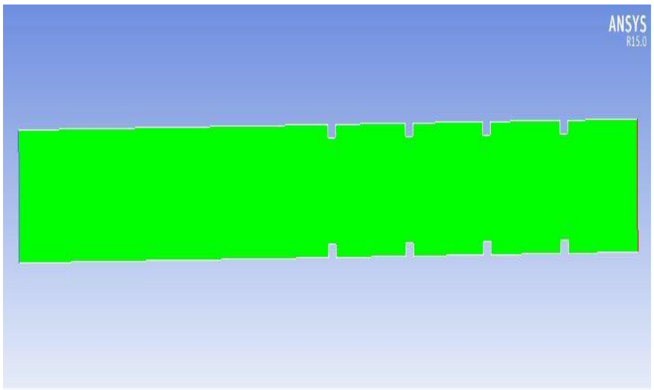

Figure: Display of mesh file

IV. METHODS OF SOLUTION:

In FLUENT 2D Double precision import geometry as mesh file of geometry .msh file will be read into FLUENT. Check geometry and verify its size and quality. Define models i.e Pressure based model and steady state simulation is done. In models select energy equation on and turbulence model K-epsilon model on. Selecting fluid as air.
k-epsilon model: K-epsilon (k-ε) turbulence model is the most widely recognized model utilized as a part of Computational Fluid Dynamics (CFD) to reproduce mean stream qualities for turbulent stream conditions. It is a two comparison model which gives a general depiction of turbulence by method for two transport mathematical statements (PDEs). The first driving force for the K-epsilon model was to enhance the mixing length model, and to locate a different option for mathematically recommending turbulent length scales in moderate to high many-sided quality stream.

For turbulent kinetic energy k:

$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial(\rho k u_i)}{\partial x_i}=\frac{\partial}{\partial x_j}\left[\frac{\mu_t}{\sigma_k}\frac{\partial k}{\partial x_j}\right]+2\mu_t E_{ij}E_{ij}-\rho\epsilon$$

For dissipation ε:
U_i shows velocity component in its corresponding direction .

IV. BOUNDARY CONDITIONS:

Velocity inlet- magnitude of Reynolds number of 20000 and 30000.Operating pressure as 1 atm i.e 101325 psa. Wall heat flux 2.777 w/m^2. Pressure velocity coupling SIMPLE scheme. Pressure and momentum as second order upwind. Set monitors residuals as 1e-06. Initialization of solution is done from inlet zone. Run calculation for 10000 iterations.

V. RESULTS:

Contours of Velocity profile:
Fluid with 20000 Reynolds number has been sent through the inlet it shows how the velocity changes in the duct. In the start inlet the velocity will be less and velocity becomes high in this

case at the ribbed wall the no slip condition occurs i.e. velocity will be zero at the wall. When flows gets in contact with square rib turbulators behind the ribs turbulence in flow occurs which in fact gets in contact with hot wall and the convection heat transfer occurs due to contact between cool fluid and hot wall.

$$\frac{\partial(\rho\epsilon)}{\partial t}+\frac{\partial(\rho\epsilon u_i)}{\partial x_i}=\frac{\partial}{\partial x_j}\left[\frac{\mu_t}{\sigma_\epsilon}\frac{\partial\epsilon}{\partial x_j}\right]+C_{1\epsilon}\frac{\epsilon}{k}2\mu_t E_{ij}E_{ij}-C_{2\epsilon}\rho\frac{\epsilon^2}{k}$$

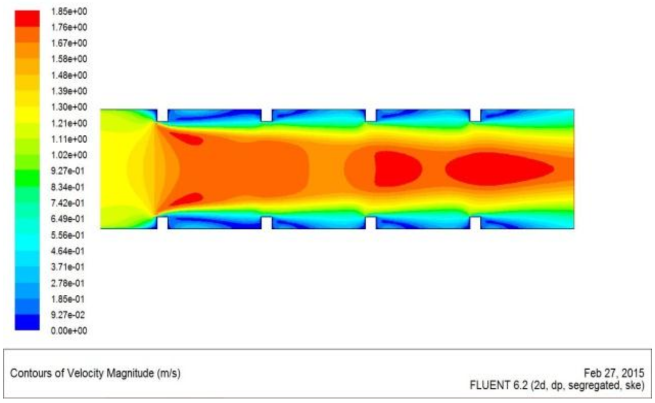

Figure: Reynolds number 20000

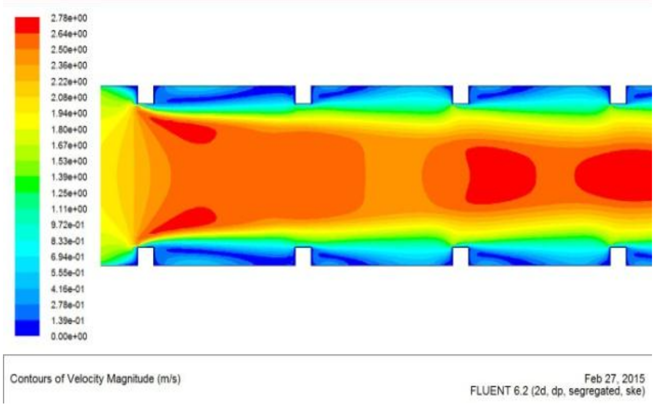

Figure: Reynolds number 30000

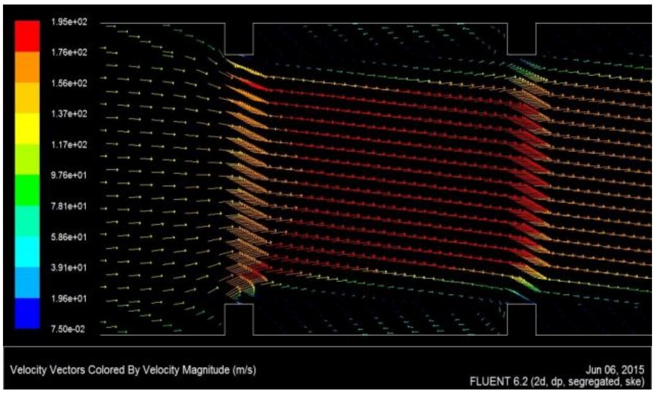

Figure: Velocity vectors between ribs showing turbulence.

Contours of Pressure:

Generally pressure drop occurs in the duct. At the inlet pressure will be very high and slowly coming down and flow getting obstructed with ribs this pressure gets decreases at the end. Fig clearly displays the pressure drop in the channel.

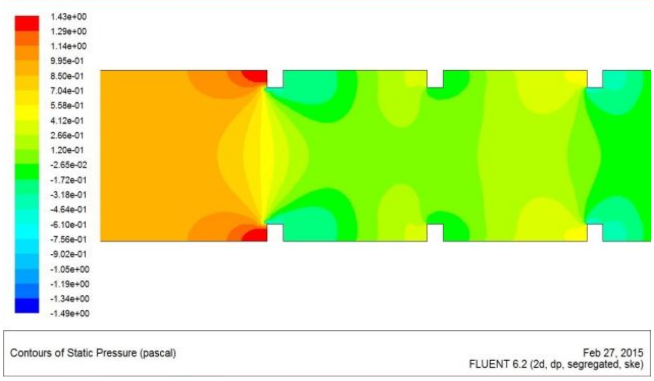

Figure: Pressure contour for Reynolds number 20000

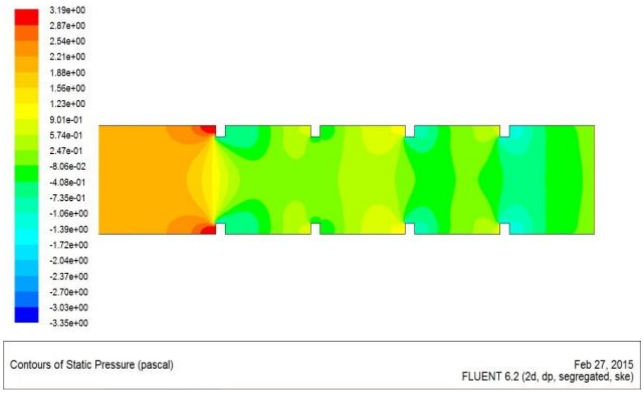

Figure: Pressure contour for Reynolds number 30000

Contours of Temperature and Heat flux:

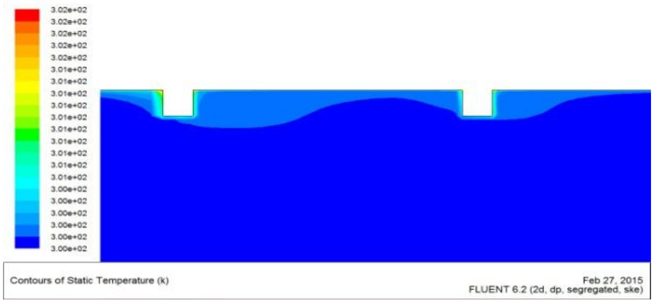

Figure: Temperature change at ribs for Re 20000

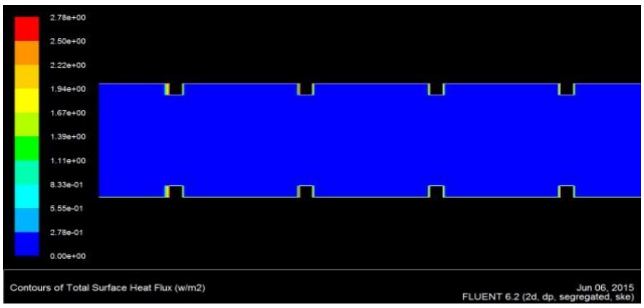

Figure: Total surface heat flux at walls

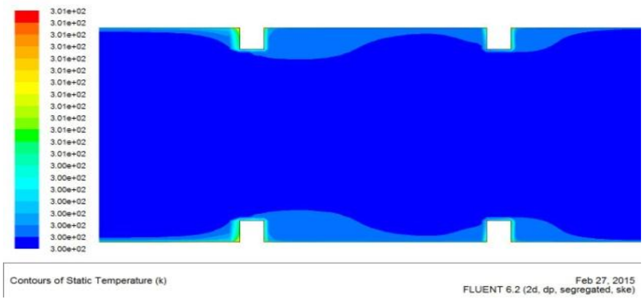

Figure: Temperature contour for Reynolds number 30000

VI. DISCUSSIONS:

From the above simulation work for 2D Internal cooling of turbine blade duct with square rib turbuators for Reynolds number 20000 and 30000 has been performed using the material medium as coolant air with 298 Kelvin temperature and the velocity variations for both Reynolds number has been discussed also the turbulence caused in between the ribs is clearly shown. Pressure drop contours, temperature and total surface heat flux contours have been shown for the simulation. Convective heat transfer occurs between fluid passing inside duct and hot walls due to turbulence of flow.

VII. FUTURE WORK:

1. Simulation to be performed for higher Reynolds numbers like 50000, 60000.
2. For different p/e ratios like 4,6 and 8.
3. Various rib turbulators geometries.

REFERENCES

[1]. Numerical simulation of gas turbine blade cooling for enhancement of heat transfer of the blade tip by Sushil sunil gaikwad, C. R. Sonawane.

[2]Turbulence Modeling for Internal Cooling of Gas-Turbine Blades by JONAS BREDBERG.

[3]. Recent progress in the computation of flow and heat transfer in internal cooling passages of turbine blades by H. Iacovides , M. Raisee Department of Mechanical Engineering, UMIST.

[4]. Reynolds averaged simulation of flow and heat transfer in ribbed ducts by A. Ooi a,*, G. Iaccarino b, P.A. Durbin c, M. Behnia.

[5]. Review on Heat Transfer Augmentation Techniques: Application in Gas Turbine Blade Internal Cooling by S. Gupta1,*, A. Chaube1 and P. Verma.

[6]. Aspect ratio effect on heat transfer in rotating two-pass rectangular channels with smooth walls and ribbed walls by WEN-LUNG FU.

[7]. Effect of rib spacing on heat transfer and friction in a rotating two-pass rectangular (ar=1:2) channel by YAO -HSIEN LIU