

# Heat Transfer Enhancement of rectangular channel with internal ribs of V Shape and $\wedge$ Shape of same cross-sectional area using CFD

Amit Mehra<sup>1</sup>, Prof. Kamal Kumar Jain<sup>2</sup>, Asst. Prof. Prabhmeet Singh<sup>3</sup>

<sup>1</sup> PG Student ME Heat Power SRIT JBP Mp 482001

<sup>2</sup> HOD Mechanical Dept. SRIT JBP Mp 482001

<sup>3</sup> Assistant Professor Mechanical Dept. SRIT JBP Mp 482001

\*\*\*

**Abstract** - The present study investigates the three-dimensional CFD simulations to investigate heat transfer and fluid flow characteristics of artificially roughened rectangular channel using Ansys- CFX. Heat transfer characteristics of the rectangular channel are investigated for Reynolds numbers ranging from 8000 to 18000 with increment of 2000. Model geometry is designed in CATIA V5 R20, and then meshed, analyzed, and post-processed using Ansys-CFX software. Fluid flow and heat transfer characteristics of different roughness configurations are simulated and then results are compared. Rectangular duct has an aspect ratio of 5, while the domain length for numerical analysis is kept 550 mm long the hydrodynamic diameter of duct is 66 mm the relative roughness height ( $e/D_h$ ) is 0.030, relative roughness pitch ( $p/e$ ) is 10. Adjacent to the bottom wall of the rectangular channel a plate of aluminium is provided whose surface is roughened by discrete rib with having some rib attack angle and the other surface of the aluminium plate is kept at uniform heat flux  $1000 \text{ W/m}^2$ . Different models of rectangular channel have been created with changes in the ribs configuration in aluminium plate. In this study three different types of ribs cross section shapes cases in aluminium plate is taken and in such a way that the cross sectional area v shape and  $\wedge$  same in every models. Those ribs cross-sectional shapes are Square Shaped, Trapezoidal Shaped From the analysis it is found that there is a significant change in heat transfer rate, friction factor, Nusselt Number and thermo hydraulic performance factor, and it is said that Vshape with square shaped profile shows the best result. Likewise in case of  $\wedge$  shape best result is obtained for square shaped profile as compared and Right Angled Shaped and every different ribs cross-section shapes models are analyzed at an attack angle of  $35^\circ$ . to the other two profile

**Keywords:** Computational Fluid Dynamic (CFD), Trapezoidal Shaped, Square Shaped

## 1. INTRODUCTION

Gas turbines play a vital role in today's industrial environment. As the demand for power and energy increases, improvement in power output and thermal efficiency of gas turbine is essential, this can be achieved by efficient cooling methods. The cooling of gas turbine components using internal convective flow of a single phase gas has been used for last 50 years; from simple smooth

cooling passage to very complex geometries involving many different surfaces, architectures and fluid-surface interaction. The main goal is to obtain high overall cooling effectiveness with lowest possible penalty on thermodynamic performance cycle. An advanced gas turbine engine operates at a high temperature of  $1500^\circ \text{C}$  to improve thermal efficiency and power output. This high temperature at rotor blade can exceed the melting temperature of the metal. It is mandatory for the blades and vanes to be cooled so that they withstand these high temperatures. 20% to 30% of the compressed air at  $650^\circ \text{C}$  is extracted from the compressor and passed through the high pressure turbine. With cool compressed air, the blade temperature can be lowered to approximately  $1000^\circ \text{C}$ , which is needed for safe operation of the engine. For these high temperature turbine blades, cooling methods require key and innovative technologies. The metal temperature of the turbine cooled vanes and blades should be predicted in design stages as accurately as possible, to reduce the period of product development cycle and to accurately predict the life of blades and vanes; as these are dependent on the metal temperature. The life of turbine blades is dependent upon accurate mapping of blade surface temperature, local heat transfer coefficients and prevention of local hot spots. Figure 1.1 shows the cross sectional view of the turbine vane and blade. Complex flow around the vanes and blades makes prediction of metal temperature difficult. The trend of heat flux is similar in both vane and blade, on suction side the flow transition from laminar to turbulent increases the heat transfer coefficient flux. Similar change occurs on the pressure side. Due to high velocity and complex flow around the gas turbine, it is important to obtain data that will help in designing efficient cooling technologies. Detailed hot gas path heat transfer distribution and film cooling data are needed for airfoil. A stator vane surface heat transfer is affected by combustor developed high turbulence, laminar to turbulent transition, acceleration, film cooling flow, platform secondary flow and surface roughness. These factors as well as the rotational, centrifugal forces

## 2. METHODOLOGY AND CFD GOVERNING EQUATION

As discussed earlier that the cooling technique is must in gas turbine blades or vanes. As the hot gases passes over the turbine blade may increase the profile temperature of

turbine blade at such extent that it can results in melting of the turbine blade. So the blade had been provided with cooling passage (duct) so that the cold air passes through these ducts in order to cool the blade.

So in this study a rectangular channel has been considered of rectangular section 200x40 mm<sup>2</sup> with length of duct is 550 mm. The bottom surface of rectangular channel is provided with a plate of aluminium which is having number of ribs along the surface of rectangular channel, Along with 3 different ribs cross section shapes in such a way that the cross section area of every shape is same in every individual model, and the bottom surface of plate is kept under uniform heat flux of 1000W/m<sup>2</sup>. The air has been passed through duct at different Reynolds number range from 8000 to 18000 with increment of 2000. Following parameters have been studied in this study.

1. To find out the Friction factor in all the cases for the Reynolds numbers ranging from 8000-18000 at the increment of 2000.
2. To find out the Nusselt Number in all the cases for the Reynolds numbers ranging from 8000-18000 at the increment of 2000.
3. To find out the ratio of Nusselt Number for ribbed channel to Nusselt Number for smooth channel for the Reynolds numbers ranges form 8000-18000 at the increment of 2000.
4. To find out the ratio of Friction factor for ribbed channel to Friction factor for smooth channel) for the Reynolds numbers ranges form 8000-18000 at the increment of 2000.
5. To find out the Thermo hydraulic Performance factor (whose value depends upon the value of nusselt ratio and friction ratio and has been discussed in this chapter later) of different ribs cross section in case of V shape and ^ shape for the Reynolds numbers ranges form 8000-18000 at the increment of 2000.

The model of all the cases has been modelled in 3D modelling software CATIA V5-R20 version and then the model of all the cases has been saved in the format of .stp. These .stp files are than import in the ANSYS CFX module for analysis.

Later in this chapter it has been discussed that what methodology has to be adopted in ANSYS CFX module in order to complete the analysis.

With reference to above discussion ANSYS CFX used the computational fluid dynamics (CFD) approach to analyse system involving fluid flow, heat transfer and phenomena such as chemical reaction a set of mathematical equations is used to construct a numerical model first which describe the flow in order to obtain the flow variables through the flow

domain .These equations are then solved using a computer program.

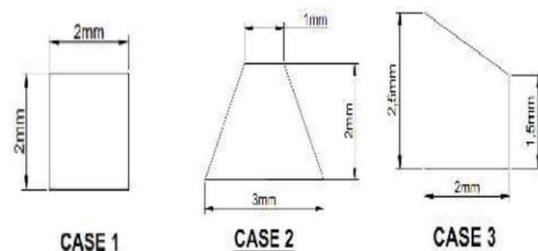
Since the arrival of the digital computer, CFD has received extensive attention and has been widely used to study various aspects of fluid dynamics. The development and application of CFD have undergone considerable growth, and as a result it has become a powerful tool in the design and analysis of engineering and other processes. The vital aspect of CFD analysis is solving of Navier-Stokes equations accurately, reliably and quickly. ANSYS CFX software is above all other CFD approaches by virtue of its latest solving technology .The use of ANSYS translates into significant and quite predictable. It is experience that predictability and efficient use of CFD helps in design options optimization systems

Before you begin to format your paper, first write and save the content as a separate text file. Keep your text and graphic files separate until after the text has been formatted and styled. Do not use hard tabs, and limit use of hard returns to only one return at the end of a paragraph. Do not add any kind of pagination anywhere in the paper. Do not number text heads-the template will do that for you.

Finally, complete content and organizational editing before formatting. Please take note of the following items when proofreading spelling and grammar:

### 3. PHYSICAL MODELS

In this study rectangular channel with length (L) of 550 mm width(W) of 200 mm and height (H) of 20 mm is taken in analysis. Hydraulic diameter (Dh) for the channel is 66 mm respectively. bottom of the rectangular channel is provided with a plate having number of ribs. The ribs attack angle for every individual model is taken as 35°. With zig-zag gapping has been provided in the rib path. Here in this study the cross section of rib has been changed and also the attack angle  $\alpha$  is 35°. There are total three cross section of rib has been modelled in such a way that the cross section area of every case is same with three different rib attack angle of 35°, 45° and 55° below is the figure of all the three cross sections and it also shows that the every case has 4mm<sup>2</sup> area.



$$A1=2 \times 2=4 \text{ mm}^2 \quad A2=(3+1)/2 \times 2=4 \text{ mm}^2$$

$$A3=[(2 \times 1.5)+(1 \times 2/2)]=4 \text{ mm}^2$$

The name of above three rib cross-section shapes is

CASE1 - Square shaped

CASE 2- Isosceles trapazoidal shape

CASE 3- Right anlge trapozoidal shape.

#### 4. MATERIAL PROPERTIES

This section of the input contains the options for the materials to be chosen. Air is passing in the rectangular duct under constant wall Heat Flux of 1000 W/m<sup>2</sup> condition in the present work. Under materials Tab in CFX, fluid considered is Air -liquid. Solid (plate wall) material considered for analysis is Aluminum. The selection of material is done. Material selected is air. The property of air and aluminium is taken as follows.

**Table.5.1. PROPERTIES OF AIR**

Sr. No.	PROPERTIES	VALUE
1	viscosity ( $\mu$ ) kg/ms	1.7894*10 <sup>5</sup>
2	Specific heat capacity (Cp) j/kg k	1006.43
3	Density ( $\rho$ ) kg/m <sup>3</sup>	1.225
4	Thermal conductivity (k)	0.0242

**Table -5.2: PROPERTIES OF ALUMINIUM**

Sr. No.	PROPERTIES	VALUE
1	Specific heat capacity	1006.43
2	Density	1.225
3	Thermal conductivity	0.0242

#### 5. BOUNDARY CONDITION

Boundary conditions are a set of properties or conditions on surfaces of domains, and are required to fully define the flow simulation. The outer wall of the bottom plate was assumed to be perfectly smooth with zero roughness height. A constant wall heat flux of 1000 W/m<sup>2</sup> was used at the wall boundary. Velocity at inlet section of air is taken in the range of 1.771 to 3.98 m/s. One face of the air domain along the length of domain, an interface between air and aluminium plate with surface roughened by ribs is considered and other three faces of air domain.

#### 6. RESULTS AND DISCUSSIONS

This chapter deals with the numerical analysis of air flowing through a rectangular channel with bottom part of rectangular channel is provided with aluminum plate and at one end of the plate internal ribs are provided with different rib attack angle and different rib shape cases of same cross section area. Another end of the plate is provided with uniform heat flux of 1000 W/m<sup>2</sup> and then comparative analysis has been done for different cases and the following parameters has been adopted for comparisons are Nusselt

Number, Nusselt Ratio (i.e Nusselt Number for ribbed channel/Nusselt Number for smooth channel), Friction factor, Friction Factor Ratio (i.e friction factor or ribbed channel/ friction factor or smooth channel) and Thermo Hydraulic Performance Factor.

#### 7. FUTURE SCOPE

In future the study can be made for W shape rib profile AND M with same ribs cross sections or with different ribs cross sections. In future the same study can be done using different rib attack angles. Here in this study bottom plate is considered as aluminum in future different material can be taken in consideration like copper, steel etc Here in this study the roughness is provided by adding ribs in the aluminum plate. In future the grooves can also be provided in the aluminum plate in order to increase roughness in surface, so that further heat transfer enhancement can be achieved.

#### REFERENCES

- a).Heat transfer enhancement of Gas turbine blades cooling rectangular channel with internal ribs of different rib cross sections using CFD. Deepak Kumar Patel ,Prof Kamal Kumar Jain ,Dr.R K Dave , International Journal of Engineering Research & Technology (IJERT) ISSN: 2278-0181 (IRJET) VOL 03 ISSUE 05 MAY 16
- b). Sandeep Kumar Karole, Ishwar Singh," Heat Transfer And Friction Behaviors In Rectangular Duct With Two Different Ribs". International Journal of Science Technology & Engineering | Volume 1 | Issue 9 | March 2015 ISSN (online): 2349-784X.
- c).RahulSaikhedkar,YogeshTembhurne."Computational Analysis Of Effect of Pitch on Heat transfer Performance In Duct". International Journal of Emerging Technologies in Computational and Applied Sciences, 12(3), March- may, 2015, pp. 290-294.d).Arkan Al-taie, Hasasn Ali Jurmut." Experimental and Numerical investigation of convective heat Transfer In A Circular Tube With Internal Square Ribs". Journal of Babylon University/Engineering Science/No. 4/Vol. (22):2014.
- e).Linqi Shui, Jianmin Gao, Xiaojun Shi, Jiazeng Liu and Liang Xu." The Ffect Of Cooling Conditions On Convective Heat Transfer And Flow In A Steam-Cooled Ribbed Duct". Journal of Mechanical Science and Technology 28 (1) (2014) 331~341.
- f).AbhilashKumarR.Saravana Sathiya Prabhahar,"Numerical Investigation Of Heat Transfer Characteristics In A Square Duct With Internal Ribs". International Journal of Innovative Research in Science, Engineering and Technology Volume 3, Special Issue 1, February 2014.