International Journal of Research
Available at https://edupediapublications.org/journals

e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 04 Issue 10
September 2017

Heat Transfer Analysis From A Concave Surface Using Multiple Circular Jet Impingement With The Influence Of Turbulence Model

Gomasu Sreenu, & Ms. S. Sushma.
[1]M.Tech in THERMAL ENGINEERING from Ellenki college of Engineering and Technology, JNTU, Hyderabad, Telangana, India.

[2]**Associate Professor,** Ellenki college of Engineering and Technology, JNTU, Hyderabad, Telangana, India

ABSTRACT

A 5 x 5 inline array of circular jets of aspect ratios [h/d=1 & 1.5] is considered in this project, to make impingement on a simulated gas turbine leading edge. The leading edge is simulated as a semi cylindrical concave surface. The jets are emerging from the holes drilled in a fluid plenum, to which fluid is supplied. A constant surface temperature is assumed over the leading edge (concave surface) and the heat transfer studies are performed, analytically. The flow is turbulent in nature in the space between the jet exit and the impingement plate. Three fluids are considered for analysis and compared. The fluids are air, CO_2 and Helium. 2D models of the turbine blade with jet impingement is done in Creo 2.0. CFD Fluent analysis is done in Ansys. Analysis is carried out at different Reynolds number 8000, 12000 & 14000.

INTRODUCTION

Impinging jets provide an effective and flexible way to transfer energy or mass in industrial applications. A directed liquid or gaseous flow released against a surface can efficiently transfer large amounts of thermal energy or mass between the surface and the fluid.

LITERATURE SURVEY

In the paper by N. Zuckerman and N. Lior[1], the applications, physics of the flow and heat transfer phenomena, available empirical correlations and values they predict, and numerical simulation techniques and results of impinging jet devices for heat transfer are described. The relative strengths and drawbacks of the k–e, k–o, Reynolds stress model, algebraic stress models, shear stress transport, and v2f turbulence models for impinging jet flow and heat transfer are compared. Select model equations are provided as well as quantitative assessments of model errors and judgments of model suitability.In the paper by Abdulla R. Al Ali. Isam Janajreh[2], Various industrial applications use jet impingement against surface to provide an effective mode of heat transfer. Its application includes, but not limited to, heat treatment, thermal management of optical surfaces for defogging, cooling of critical machinery structures, and rocket launcher cooling. In this study, numerical analysis of various heat transfer configurations of jet impingement on a semi-circular surface is carried out. These configurations were compared on the basis of effective heat transfer by achieving higher Nusselt number and lower surface temperature as convection heat is becoming the dominant phenomenon. The numerical model was developed for considering the application of a uniform heat flux on a curved surface subjected to jet flow that simulating an internal channel under cooling. The results found to be in agreement with the literature experimental data. To gain more insight on the underlining physics of the flow, a sensitivity analysis on the jet impingement configuration and flow conditions were conducted and was demonstrated to the inner cooling of the 1st stage gas turbine blade.

3D MODELING AND ANALYSIS OF CONCAVE SURFACE OF GAS TURBINE BLADE WITH JET IMPINGEMENT

The reference journal for this project N. Zuckerman and N. Lior, Jet Impingement Heat Transfer: Physics, Correlations, and Numerical Modeling, specified as [1] in references chapter.

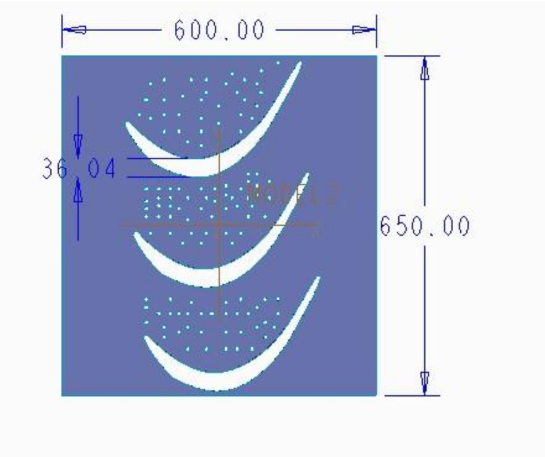


Fig – 2D Surface model for H/D = 1

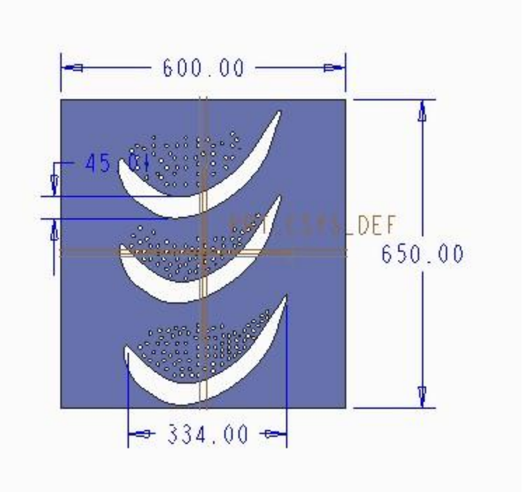


Fig – 2D Surface model for H/D = 1.5

VELOCITY CALCULATIONS FOR DIFFERENT FLUIDS AT DIFFERENT REYNOLDS NUMBER

$Re=\rho vl/\mu$

Re = Reynolds number

ρ = Density of fluid (Kg/m^3)

v = velocity (m/s)

l = length of the surface = 334mm

μ = Viscosity (Kg m/s)

CFD ANALYSIS OF JET IMPINGEMENT ON CONCAVE SURFACE OF GAS TURBINE BLADE

CFD analysis is done by varying H/D ratios where H = Height of nozzle and D = Diameter of nozzle.

H/D – 1

FLUID - AIR

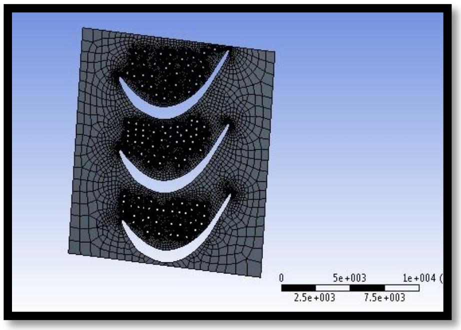
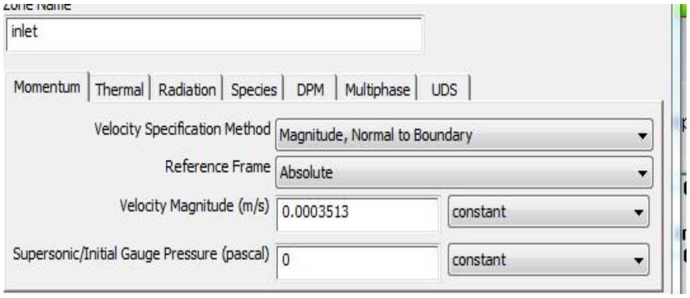


Fig:- Meshed model with H/D = 1

REYNOLDS NUMBER R_e=8000

Boundary conditions → select air inlet → Edit → Enter Inlet Velocity → 0.0003513m/s



Zone Name
inlet

Velocity Specification Method: Magnitude, Normal to Boundary

Reference Frame: Absolute

Velocity Magnitude (m/s): 0.0003513 constant

Supersonic/Initial Gauge Pressure (pascal): 0 constant

Inlet velocity (Re=8000) for Air

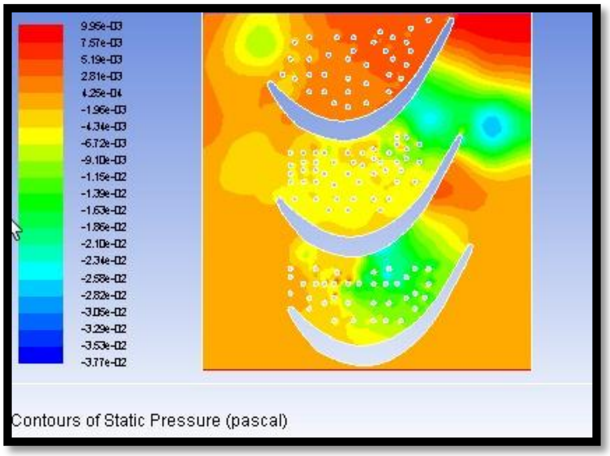


Fig. Pressure for H/D=1 at R_e = 8000 for Air

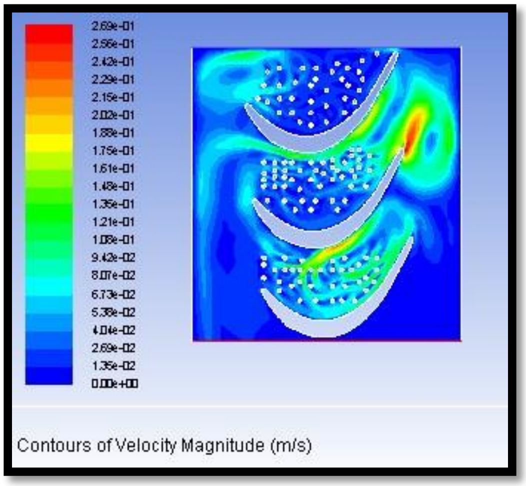


Fig. Velocity for H/D=1 at R_e = 8000 for Air

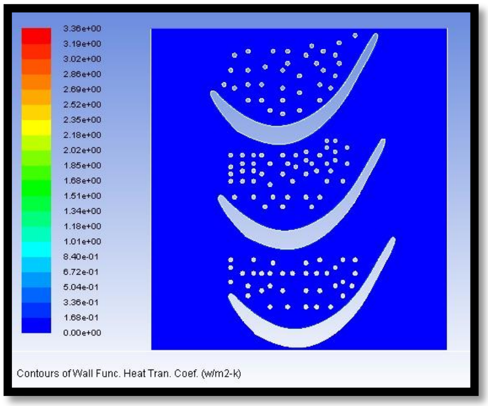
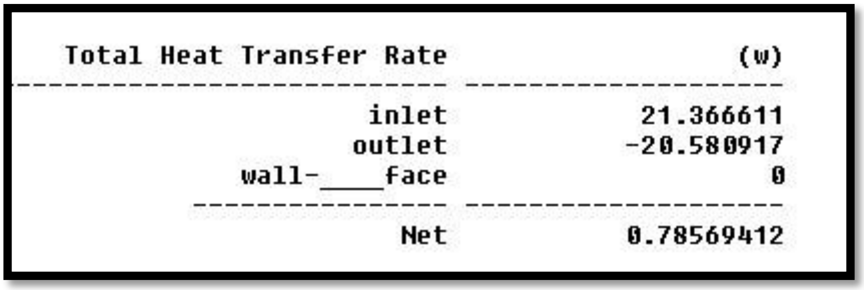


Fig. wall heat transfer coefficient H/D=1 at R_e = 8000 for Air



```
  Total Heat Transfer Rate                        (w)
-------------------------------- --------------------
                         inlet             21.366611
                        outlet            -20.580917
                   wall-____face                   0
                ---------------- --------------------
                           Net            0.78569412
```

Fig:- Total heat transfer rate (Re=8000) for Air

RESULTS & DISCUSSIONS

H/D RATIO: 1

Air

Reynolds number	**Pressure (Pa)**	**Velocity (m/s)**	**Wall heat transfer coefficient (W/m^2 K)**	**Total heat transfer rate (W)**
8000	$9.96e^{-03}$	$1.08\ e^{-01}$	3.36e+00	0.78569412
12000	$5.60\ e^{-03}$	$1.90\ e^{-01}$	3.36e+00	9.6074694
14000	$2.94\ e^{-03}$	$2.69e^{-01}$	3.36e+00	13.385399

CO_2

Reynolds number	**Pressure (Pa)**	**Velocity (m/s)**	**Wall heat transfer coefficient (W/m^2 K)**	**Total heat transfer rate (W)**
8000	$1.32\ e^{-02}$	$8.3\ e^{-02}$	2.01e+01	12.114908
12000	$1.06\ e^{-02}$	$2.27\ e^{-01}$	2.01e+01	23.089289
14000	$1.00\ e^{-02}$	$2.3\ e^{-01}$	2.01e+01	33.415934

Helium

Reynolds number	**Pressure (Pa)**	**Velocity (m/s)**	**Wall heat transfer coefficient (W/m^2 K)**	**Total heat transfer rate (W)**
8000	5.57 e^{-03}	3.49 e^{-02}	2.11e+01	1.406044
12000	6.02 e^{-04}	5.15 e^{-02}	2.11e+01	5.6766434
14000	9.11 e^{-04}	1.57 e^{-01}	2.11e+01	8.2225494

By observing pressures, the values are decreasing by increase of Reynolds number resulting from velocity gradients present at the nozzle exit. By observing velocities, the values are increasing by increase of Reynolds number. The heat transfer coefficient does not depend on the Reynolds number, it is constant. The heat transfer coefficient is more when Helium is used.By observing heat transfer rates, the values are increasing by increase of Reynolds number. Heat transfer rate is more when CO_2 is used.

H/D RATIO: 1.5

Air

Reynolds number	**Pressure (Pa)**	**Velocity (m/s)**	**Wall heat transfer coefficient (W/m^2 K)**	**Total heat transfer rate (W)**
8000	2.72e^{-05}	1.85e^{-03}	8.93e+00	0.067476273
12000	6.25 e^{-05}	2.91e^{-03}	8.93e+00	0.3694706
14000	8.66 e^{-05}	3.45 e^{-03}	8.93e+00	0.608967119

CO_2

Reynolds number	**Pressure (Pa)**	**Velocity (m/s)**	**Wall heat transfer coefficient (W/m^2 K)**	**Total heat transfer rate (W)**
8000	3.75 e^{-05}	2.21 e^{-03}	5.35e+00	0.44201279
12000	8.77 e^{-05}	3.47e^{-03}	5.35e+00	0.79247665
14000	1.22e^{-05}	4.13e^{-03}	5.35e+00	1.2823814

Helium

Reynolds number	**Pressure (Pa)**	**Velocity (m/s)**	**Wall heat transfer coefficient (W/m^2 K)**	**Total heat transfer rate (W)**
8000	5.211e^{-03}	1.85 e^{02}	5.61e+01	0.38368225
12000	6.16 e^{-04}	2.78e^{-02}	5.61e+01	2.056812
14000	4.50 e^{-04}	3.24e^{-02}	5.61e+01	3.4442902

CONCLUSION

CFD analysis is performed for different Reynolds number 8000, 12000 & 14000 in Ansys.

By observing CFD analysis results, pressures values are decreasing by increase of Reynolds number resulting from velocity gradients present at the nozzle exit. Velocities are increasing by increase of Reynolds number. The heat transfer coefficient does not depend on the Reynolds number, it is constant. The heat transfer coefficient is more when Helium is used. Heat transfer rates are increasing by increase of Reynolds number. Heat transfer rate is more when CO_2 is used for H/D is 1 and Helium is more for H/D is 1.5. By comparing values for H/D = 1 & 1.5, the pressures and velocities are decreasing by increase of H/D ratio. The heat transfer coefficient is increasing by increase of H/D ratio. Heat transfer rates are decreasing by increase of H/D ratio.

REFERENCES

[1] N. ZUCKERMAN and N. LIOR, Jet Impingement Heat Transfer: Physics, Correlations, and Numerical Modeling, Advances in Heat Transfer VOL. 39

[2] Abdulla R. Al Ali. Isam Janajreh, Numerical Simulation of Turbine Blade Cooling via Jet Impingement

[3] Kartik Jujare , Shashank Kumat, Akash Thawkar, Experimental Investigation of Jet Impingement Cooling On a Ribbed Surface with Holes

[4] NARESH R, Experimental Investigation Of Heat Transfer On An Inclined Plate Impinged With Cold Air Jet.

[5] Dr. Assim. H. Yousif, Experimental Investigation of Jet Impingement Cooling On Ribbed Target Surface

[6] Frank Haucke, Henning Kroll, Experimental Investigation of a 7 by 7 Nozzle Jet Array for Dynamic Impingement Cooling.

[7] Harikrishnan.M, Arun Jacob, Numerical And Experimental Investigation Onair Jet Impingement Cooling.

[8] G. Srivalli, Fluid Flow And Heat Transfer Analysis Of Laminar Multiple Square Jets Impinging On A Flat Plate.

[9] Ferrari, J., Lior, N., and Slycke, J. (2003). An evaluation of gas quenching of steel rings by multiple-jet impingement. J. Mater. Process. Technol. 136, 190–201.

[10] Martin, H. (1977). Heat and mass transfer between impinging gas jets and solid surfaces. Adv. Heat Transfer 13, 1–60.