iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH
IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 12 Issue: IV Month of publication: April 2024

DOI: https://doi.org/10.22214/ijraset.2024.60775

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 12 Issue IV Apr 2024- Available at www.ijraset.com

Structural Analysis & Effect of Impingement Cooling Flow in the Internal Surface on Temperature Distribution of a Vane in Gas Turbine Using Taguchi Technique

Shaheen Mahboob[1], D J Johnson[2], P. Gayatri [3]
[1]Student (MTech CAD/CAM), [2]Associate Professor, [3]Assistant Professor, Department of Mechanical Engineering, Pragati Engineering College,Kakinada, Andhra Pradesh, India

***Abstract:** Gas turbine always consider one of the most important systems in the modern engineering applications, because it has continuous ability to generate electric power. In gas turbines the major portion of performance dependency lies upon turbine blade design ,the blades are considered one of the important and expensive parts in the gas turbines , where the blades of first stage from failure .The blades of the gas turbine suffer from tensile stresses due to centrifugal forces resulting from the high rotational speed and because of the loading of densegasses at a high temperature and speed ,as the centrifugal force is one of the problems that thedesigner of the turbine blades faces , as the designer aims to reduce stresses within the permissible limit. CFD study was carried out for evaluating the performance of a utility Steam Turbine. The flow in a turbine blade passage is complex and involves understanding of energy conversion in three dimensional geometries. The performance of turbine depends on efficient energy conversion and analyzing the flow path behavior in the various componentsof Steam Turbine. This study seeks the optimization of an axial turbine from a small gas turbine engine developed by ITA using Computational Fluid Dynamics (CFD) and Multi- Objective Optimization techniques focused on geometry changes to maximize the turbine performance. The simulation process will be done through the use of the commercial software SolidWorks.*
***Keyword:** Generate Electric Power, Tensile Stresses, Centrifugal force, CFD, Turbine Engine*

I. INTRODUCTION

The aeronautical industry and its competitiveness in the market of increasingly efficient aircraft, constantly seek through a lot of research and technological development, an increasingly safe and cheap solution to the final consumer who is the passenger. These surveys can follow different goals such as: drag reduction, lift improvement, fuel economy, noise reduction, improved thermal comfort, greater ease of manufacturing, robust parts design to increase your life, increased propulsion and reduced costs generally. For each of these fields, several studies were required. CFD allows observation of flow properties at locations which may not be accessible to (or harmful for) measuring instruments. For example, inside a combustion chamber, or between turbine blades. Designers and analysts can study prototypes numerically, and then test by experimentation only those which show promise. The aerodynamics of the flow in a turbine stage is rather complex and is still the subject of many ongoing research activities in the gas turbine community. The flow isinherently three dimensional due to the blade passage geometry with features such as twisting of the blade along the span, clearance between the blade tip and the shroud, film cooling holes and end wall contouring.

The Modern updates in computational techniques have given scope to understand the behavior of flow of fluid before the experimentation. Computational Fluid Dynamics (CFD) is one of the finest techniques to Check the behavior of flow over an object. Studies have proven CFD has given close results to experimental analysis [1,2]. Use of Navier stokes equation in CFD solving of the fluid Domains gives a better result with less error from the experimental analysis [3]. So, the study is conducted by modeling and simulation technique. The main factor that typically controls the performance of a gas turbine is the profile of the van. Several studies are been conducted to study the response of vane at different profiles. The chamber of vane plays a crucial role in selecting the profile. The increment in chamber shows a relative improvement in performance of the vane [4].

This is because the increment in chamber causes more pressure drop which leads to high drag force that helps in rotation [5]. Also, the position of maximum chamber plays a vital role in the performance of the vane. The position of maximum chamber at 30% of chord length have shown a dominant performance while studying the nature at a range of 10% to 60% [6].

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 12 Issue IV Apr 2024- Available at www.ijraset.com

The study on the SG 6043 and NACA 4412 is shown a dominant performance in SG 6043 over NACA 4412 in the enhancement of the aerodynamic characteristics [7]. The NACA 6409 has better fluid separation capacity which is proven from experimental results. But the NACA 4412 has dominated the results compared with NACA 6409[11, 27, 28]. The study on airfoil NACA 2412, NACA 4412 & NACA 6412 have proven that the profile of NACA 6412 has the superior flow separation over the remaining considered profiles [2,3]. The work is continued to NACA 6412, NACA 7412 & NACA 8412. The profile of NACA 8412 have the greater flow separation and lift-to-drag ratio [10,13]. The study of flow on RAE 2282, NACA4415, NACA 4418 and NACA 6409 have shown that the RAE 2282 has best characteristics among the selected airfoils [29].
Coming to the material and coating of profile the material with high thermal conductivity should be considered [15,25]. The thermal study between the stainless steel and aluminum with epoxy coating. It shown that the stainless steel has shown better thermal characteristics [14]. The study of material is further continued between Inconel & titanium T6 [18], Titanium aluminum alloy [20] , 617Nickel & chromium steel[16], SS 304[17], Inconel 718 &N155[19] and EN 24, AISI 4130 & ZAMAK [21]. It is shown that the ZAMAK has best characters among all the materials studied above. Coming to coating material Y2O3 [11] and Zirconia [22] have shown that the zirconia is better suitable for coating.
The impingement holes are one of the best choices for cooling of gas turbine. The vane with 7,8,9,10,11 & 12 is arranged along the chord line. The vane with 8 holes on the chord line have shown better thermal distribution among the other configurations [20]. The leading-edgetemperature and heat transfer analysis is carried out at 0, 5, 9 and 13 impingement holes. The vane with 13 holes has shown the minimum temperature at leading edge and optimum heat transfer [26]. The arrangement of the impingement also plays a viral role in heat transfer. The study of arrangement of inline and staggered holes. It is shown that 14 staggered holes have shown better heat transfer [23]. The density of coolant flowing through the impingement holes increases the efficiency of cooling [24].

II. METHODOLOGY

The gas turbine profile with RAE 2822, SG 6043 and NACA 8312 are studied. The chord of 300mm is selected. A 7^0-angle twist is applied for better improvement of lift [24]. The airfoils is applied with the inlet parameters of 8m/s at a temperature of 60^0C. Also, the surface roughness of profile is adjusted to 2 Micrometers. The airfoils are tested at 0^0–150- degree angle of attack at a interval of 2.5^0. The best airfoil is applied with the coolant flow of , 0.02, 0.03 Kg/s and the gas flow at temperature of 60^0C, 120^0C & 180^0C. The vane material of ZAMAK and coating of 0.1mm is applied with zirconia. The turbulence intensity of 2% is considered. Also it is assumed that the inlet gas has a inlet kinetic energy of 1 m^2/S^2 and specific depression of 1 S^{-1}. These results are used for Taguchi optimization.

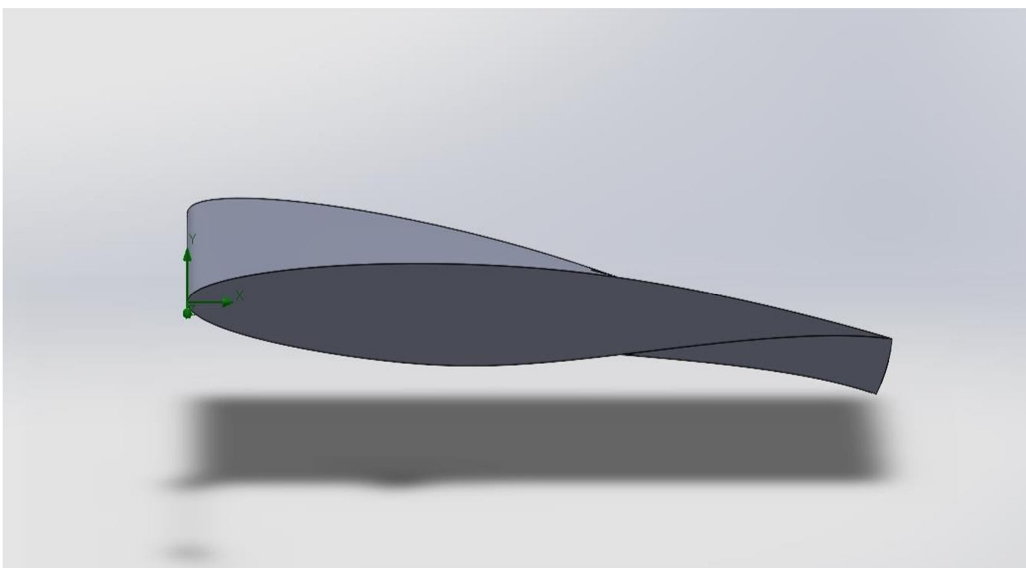


Fig-1: RAE 2822 Airfoil

The geometries are applied with principles such as the continuity principle, Bernoulli's principle and equations such as energy equation, kappa – epsilon equation. The calculation and evaluation are termed as follows:

1) Continuity Equation

The constitute of the fluid entering the intended profile must be same as the constitute of the fluid leaving the profile.

$$m_1 = m_2$$

$$\frac{dm1}{dt} = \frac{dm2}{dt}$$

$$\rho_1 A_1 U_1 = \rho_2 A_2 U_2$$

$$A_1 V_1 = A_2 V_2$$

2) Momentum Equation

The rate at which the momentum of a fluid particle changes, must be equal to theforces acting along the flow stream

F = mass × acceleration

Now,

Consider a functional sample from the depicted fluid flow

Let,

dA = cross sectional area of considered functional fluid sampledL = length of the functional fluid element

dW = weight of the functional fluid elementu = velocity of the functional fluid element p = pressure of the functional fluid element

Assume that the fluid is steady, non-viscous, and incompressible so that the frictional lossesare zero and the density of the fluid is constant

The different forces acting on the fluid are,

a) Pressure force acting in the direction of the flow (PdA)

b) pressure force acting in the opposite direction of the flow [(P+dP)dA]

c) gravity force acting in the opposite direction of the force (dWsin θ).

Therefore,

Total force = gravity force + pressure force

The pressure force is considered in the direction of low

$$Fp = P\, dA - (P + dP)dA$$

The gravity force considered in the direction of flow

$$Fg = -dw\, sin\theta \qquad [W = mg = \rho\, dA\, dL\, g]$$

$$= -\rho g\, dA\, dL\, sin\theta \quad [sin\theta = \frac{dZ}{dL}]$$

$$= -\rho g\, dA\, dZ$$

The net force is considered in the direction of flow

$$F = m\, a \qquad [m = \rho\, dA\, dL]$$

$$= \rho\, dA\, dL\, a$$

We have

(Euler’s equation of motion)

$$\frac{dP}{\rho} + u\, dU + dZ\, g = 0$$

On integrating the Euler’s equation, we get the Bernoulli’s equation

$$\int \frac{dP}{\rho} + \int U\, dU + \int dZ\, g = constant$$

$$\frac{P}{\rho} + \frac{U2}{2} + Zg = constant$$

$$\frac{\Delta P}{\rho} + \frac{\Delta U_2}{2} + \Delta Z\ g = 0 \text{ (Bernoulli's equation)}$$

3) *Energy equation:*

$$E = \square \frac{P}{\rho} + \frac{V^2}{2}$$

The resulted airfoil is applied with a rpm of 5000 to check the structural stability of the vane.

III. RESULTS AND DISCUSSION

The boundary condition is applied to the airfoil models as specified. The airfoil results are plotted as graph between the Cl/Cd ratio and angle of attack. The results are tabulated in table1.

	AOA	cl/cd	max	min	max	min	max
RAE2822	0	4.4546	8.9415	101310	101355	333.143	333.182
	2.5	4.2186	9.0676	101300	101370	333.141	333.182
	5	3.6454	9.32058	101298	101354	333.139	333.182
	7.5	3.2203	9.10565	101295	101380	333.141	333.183
	10	3.2410	9.79131	101294	101367	333.134	333.182
	12.5	2.9080	10.153	101285	101360	333.131	333.182
	15	2.5191	10.8399	101271	101401	333.124	333.182
SG 6043	0	3.8224	9.19809	101302	101360	333.14	333.182
	2.5	4.2295	8.89946	101306	101347	333.142	333.182
	5	3.8776	9.00457	101307	101349	333.142	333.182
	7.5	3.6814	9.22036	101307	101353	333.136	333.182
	10	3.4815	9.66611	101300	101358	333.135	333.181
	12.5	3.3332	9.55452	101299	101344	333.136	333.181
	15	2.9336	9.63411	101301	101362	333.136	333.182
NACA8312	0	2.8614	9.54755	101295	101365	333.137	333.182
	2.5	3.3331	9.7537	101292	101358	333.135	333.182
	5	3.2297	9.70411	101294	101351	333.135	333.182
	7.5	3.0958	9.77731	101301	101361	333.134	333.182
	10	3.0951	9.93532	101300	101347	333.133	333.182
	12.5	2.8746	9.93991	101295	101348	333.133	333.182
	15	2.6505	9.93047	101292	101347	333.133	333.182

Table 1: CFD results from airfoils

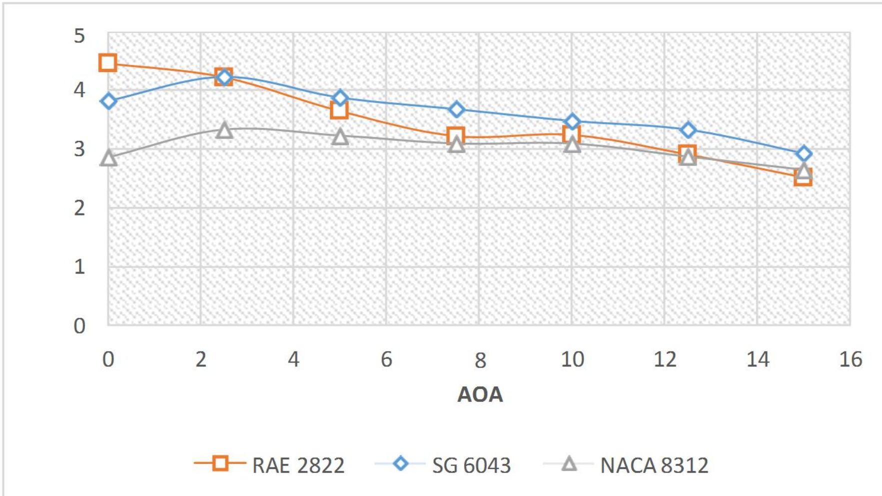


Figure 2: Cl/Cd vs Angle of attack (AOA)

From this it is seen that the RAE 2822 has a good lift to drag ratio at 0-degree angle of attack.The velocity distribution is as shown in below figure 3.

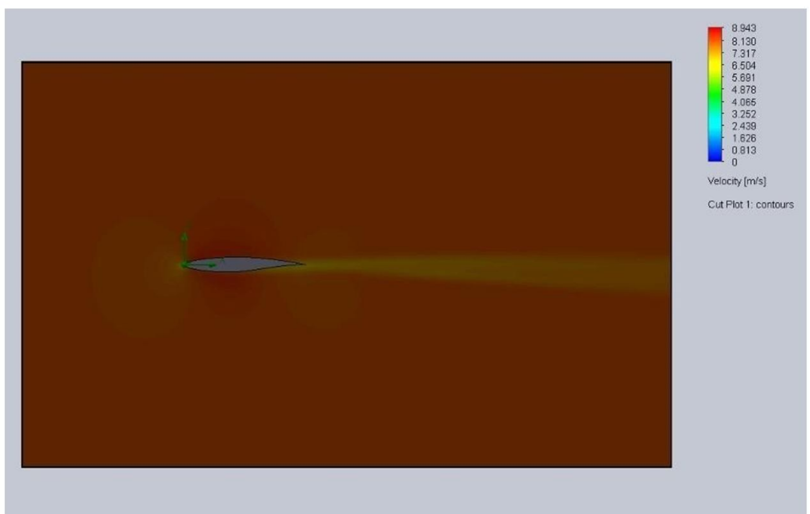


Figure 3: Velocity contours of RAE 2822 at 0^0 AOA

This RAE 2822 is applied with the air as a coolant at different mass flow rates of 0.01 to 0.03Kg/s and the inlet gas temperature in external boundary is varied as 60, 120 and 180 degrees Celsius. The heat flux in vane and flow characters are studied. The results are as follows,

S.	coolant inlet	Gas Inlet temperature	lift	drag	lift to drag	heat
1	0.01	60	0.21982	0.047969	4.582476021	15378.9
2	0.01	120	0.19973	0.044021	4.537138418	38924.9
3	0.01	180	0.18444	0.040825	4.518018283	55965.9
4	0.02	60	0.46388	0.082309	5.635871262	17501.2
5	0.02	120	0.44693	0.080009	5.586049157	44287.5
6	0.02	180	0.43489	0.078380	5.548569664	63730.7
7	0.03	60	0.87833	0.143877	6.104770047	16914.4
8	0.03	120	0.86458	0.140185	6.167485822	46644.9
9	0.03	180	0.85749	0.140253	6.113901307	67100.1

Table 2: CFD results by varying coolant flow and inlet temperature

The obtained results are been used for Taguchi level 3 optimization where coolant mass flowrate and gas inlet temperature are the input factors varying. The Lift to drag ratio and heat flux are as results. The larger is the best domain is selected. The S/N grapgh is as shown in figure 4.

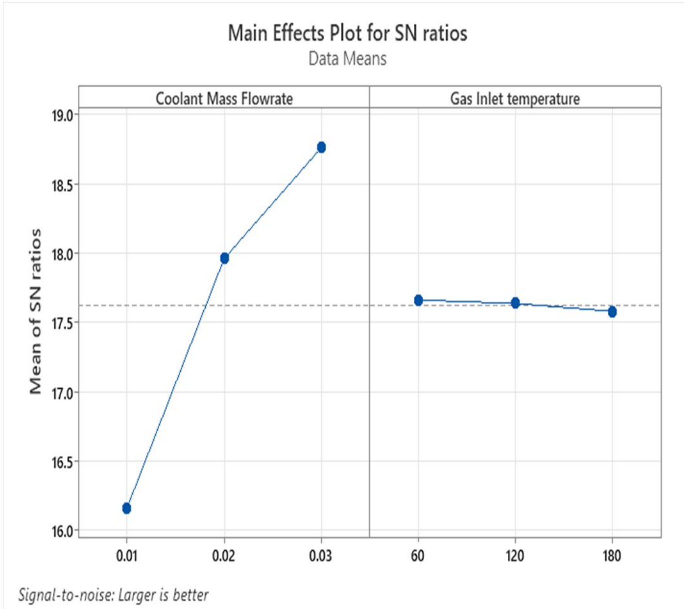
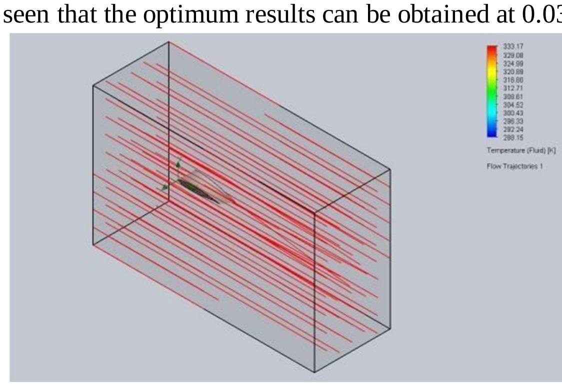


Figure 4: SN ratio graph obtained from taguchi.

It is clearly seen that the optimum results can be obtained at 0.03 Kg/s coolant inlet and 60^{0}Cof inlet gas temperature.



id T Figure 5: Fluid Temperature distribution indomain

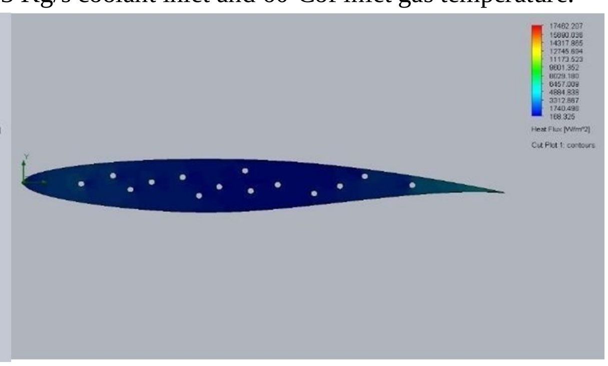


Figure 6: Heat Flux in vane

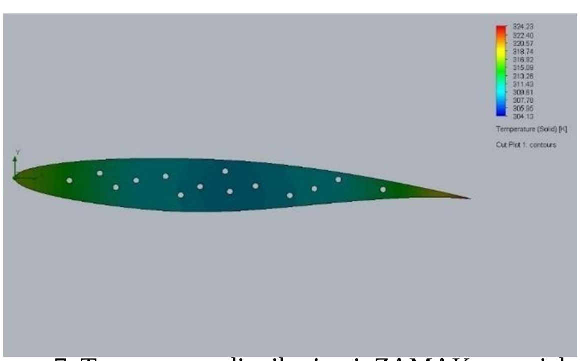


Figure 7: Temperature distribution inZAMAK material.

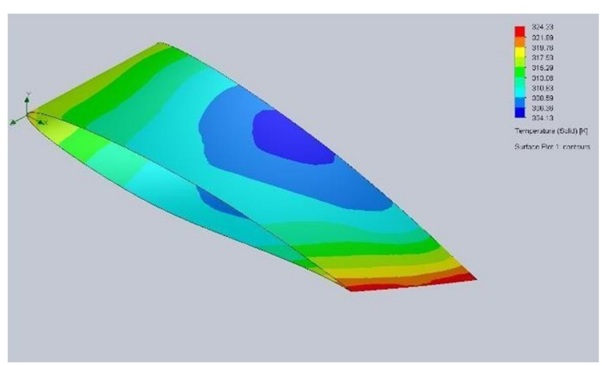


Figure 8: Surface temperature distribution onZirconia coating.

Now the model is applied with a Centrifugal force as RPM of 5100. The results are asfollows,

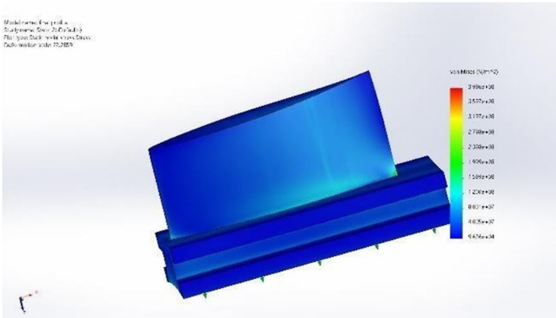


Figure 9: Stress distribution

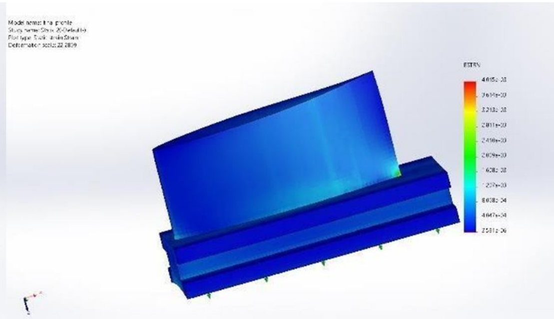


Figure 10: Strain Distribution

IV. CONCLUSIONS

The study is given various conclusions regarding flow separation and static strength of thebody. Those are:

1) The RAE 2822 airfoil has better flow separation over SAE 6043 and NACA 8312.

2) Taguchi given a optimum operation range as 0.03 Kg/s of coolant inlet and 60 degreeCelsius of gas inlet temperature.

3) RAE 2822 vane with 7^0 angle of twist has shown a good structural stiffness with stress as 3.996 e+08 N/m^2 and strain as 4.015e-03.

For future scope the profile can be further modified by means of dimples [13], etc gives abetter result.

REFERENCES

[1] Amaral LGF, Diaz, Tomita JT, et al. Optimization based on 3D CFD simulations of an axial turbine from a small gas turbine engine. Aeron Aero OpenAccess J. 2019;3(4): 198-211.

[2] Scupi A., Dinu D.: Experimental and Numerical Methods for Hydrodynamics Profiles Calculation . TransNav, the International Journal on Marine Navigation and safety of Sea Transportation, Vol.5, No.4, pp.483-486,2011.

[3] Tulus et al 2019 J. Phys.: Conf. Ser. 1376 012003

[4] Loya, A., Khan, M.Z.U., Bhutta, R.A. and Saeed, M. (2016) Dependency of Torque on Aerofoil camber Variation in Vertical Axis Wind Turbine. World Journal of Mechanics , 6, 472-486.

[5] Zubin Zaheer et al 2019 J. Phys.: Conf. Ser. 1355 012006

[6] Jithendra Sai Raja Chada et al. Study of flow by varying the position of maximum chamber thicknesson aerofoil. Technical Journal of Daukeyev University. 2021; 1(2): 69-75.

[7] Aung, Y. Y. H., Soe, M. M., & Thu, A. M. (2016). Effect of Attack angle on Aerodynamics Analysisof Different Wind Turbine Wings using Numerical Simulation. American Scientific Research Journal for Engineering, Technology, and Sciences, 26(4), 319–329.

[8] Abid, muhammad ali raza & Sarwar, M. & Tahir, Ahmad & Shah, Masood & Manzoor, S.. (2012). Gasturbine blade flow analysis comparison using CFD and Wind Tunnel. Proceedings of 2012 9th International Bhurban Conference on Applied Sciences and Technology, IBCAST 2012.

[9] Singh A, Pujari AK, Prasad BVSSS. Conjugate heat transfer analysis of a rotor blade with coolant channels. Heat Transfer. 2021;1-31.

[10] Geeri, Satish, Akhil Yuvaraj Manda, Jithendra Sai Raja Chada, and Sambhu Prasad Surapaneni. “Flow Behaviour on Aerofoils Using CFD.” Journal of Mechanical Engineering, Automation and Control Systems 1, no. 1 (June 30, 2020): 26–36.

[11] Bathineedi Nagababu, V.S.Nagi Reddy, Abdul Arif et al. Design and cfd analysis on coated steam turbine blade. 2nd International conference on recent developments in science, humanities & management, 2018.

[12] Rasal, S and Mr. Rohan R. Katwate. “Numerical Analysis of Lift & Drag Performance of NACA0012 Wind Turbine Aerofoil.” IRJET, 2017; 4(6): 2892-2896.

[13] S. . Geeri, S. P. Surapaneni, J. S. R. Chada, and A. Y. Manda, “Studies Based on CFD Behaviour of Aerofoil and Regression Analysis”, Engineering Science & Technology, vol. 3, no. 1, pp. 55–68,

[14] Oct. 2021

[15] Hao, J., Zhang, M. & Huang, X. Experimental Study on Influences of Surface Materials on CavitationFlow Around Hydrofoils. Chin. J. Mech. Eng. 32, 45 (2019).

[16] Rao, Nageswara. 2011. ‘Materials for Gas Turbines – An Overview’. Advances in Gas TurbineTechnology. InTech.

[17] [16]B.Sai Venkata Krishna, .Chakradhar Goud et al. CFD And Heat Transfer Analysis Of Air Craft GasTurbine Blade, IJRAT 2019.

[18] Kukutla, P.R., Prasad, B.V.S.S.S. Secondary flow visualization on stagnation row of a combined impingement and film cooled high-pressure gas turbine nozzle guide vane using PIV technique. J Vis 20, 817–832 (2017).

[19] Theju V, Uday P S, PLV Gopinath Reddy, C.J.Manjunath et al. Design and Analysis of Gas Turbine Blade. IJIRSET, 3(6): 2014.

[20] R D V Prasad1, G Narasa Raju2, M S SSrinivasa Rao3, N Vasudeva Rao. Steady State Thermal & Structural Analysis Of Gas Turbine Blade Cooling System. IJERT; 2(1): 2013.

[21] Deepanraj, B., & Lawrence, P. (2011). Theoretical Analysis of Gas Turbine Blades by Finite Element Method. Journal of the Institute of Engineering, 8(1-2), 1–11.

[22] G Nagendra Krishna, K.Rajesh, A.Swarna Kumari et al. Design and analysis of steam turbine bladeand shaft assembly. IJRI; 2(4): 2015.

[23] Leandro Augusto de Souza, Elisângela Martins Leal, Adilson Rodrigues da Costa & Milton Sergio Ernandes d Lima et al. Global Journal of Researches in Engineering (A) Volume XIxX Issue III Version I, 2019.

[24] Priyanka Singh, O P Shukla et al. Heat Transfer Analysis of Gas Turbine Rotor Blade Through Staggered Holes Using CFD. International Journal of Engineering Research and General Science. 4(2),2016.

[25] Sahu, M., Shaik, M. (2022). 'Numerical investigation on cooling performance of a twisted gas turbine blade with leading edge cooling holes', Journal of Computational & Applied Research in Mechanical Engineering (JCARME), 11(2), pp. 397-407.

[26] Pujari, A., Prasad, B. and Sitaram, N. (2021) Effect of Thermal Conductivity on Nozzle Guide Vane Internal Surface Temperature Distribution . International Journal of Turbo & Jet-Engines, Vol. 38 (Issue 2), pp. 135-151.

[27] K Hari Brahmaiah , M.Lava Kumar at al. (2014) Heat Transfer Analysis of Gas Turbine Blade Through Cooling Holes. International Journal of Computational Engineering Research, 4(7).

[28] Nirav Prajapati, Umang Patdiwala, Piyushkumar Surani et al .,Experimental Validation Of Naca 6409 Airfoil Using Cfd Analysis. JETIR 8(4). 2021.

[29] MD. Safayet Hossain, Muhammad Ferdous Raiyan, Mohammed Nasir Uddin Akanda, Nahed Hassan Jony. A comparative flow analysis of NACA 6409 and NACA 4412 aerofoil. International Journal of Research in Engineering and Technology. 3(10). 2014.

[30] Kushal Gendcha1 Kalpit P. Kaurase, Analysis of Transonic Flow over Airfoil using CFD for Gas Turbine Blades. International Journal for Scientific Research & Development. 4(5). 2016.

[31] Yixuan Xia, Analysis of the research status of stationary gas turbine and its material selection. Proceedings of the 3rd International Conference on Signal Processing and Machine Learning. (2023).

[32] Mohammed Talal Mohammed, Ziad Shakeeb Al Sarraf and Sabah Mohammed Jamil. Finite Element Simulation and Stress Analysis of Gas Turbine Blade Due to Centrifugal Force. International Journal of Advanced Natural Sciences and Engineering Researches. 7(6). 2023.