International Journal of Trend in Scientific Research and Development (IJTSRD)
Volume 9 Issue 4, Jul-Aug 2025 Available Online: www.ijtsrd.com e-ISSN: 2456 – 6470

Comparative Study of Kaplan Turbine Blades for Making More Rotations

Sundhara Vadivel R

Lecturer, Department of Mechanical Engineering, Sankar Polytechnic College, Sankarnagar, Tamil Nadu, India

ABSTRACT

The main objective of this project is the comparative study of the kaplan turbine blades for make the more rotations, kaplan turbines are used in hydro power plants for generating power, in this current research implementing the blade holes similar to the blade profiles, this new model flow performance can be compared with the existing blade profiles, the flow performances can be analyze through CFD Methodology, CATIA V5 software is used for creating 3D model of Kaplan Turbine and ANSYS Fluent software is used CFD Analysis.

KEYWORDS: *CFD, Kaplan turbines, CATIA, Flow analysis.*

How to cite this paper: Sundhara Vadivel R "Comparative Study of Kaplan Turbine Blades for Making More Rotations" Published in International Journal of Trend in Scientific Research and Development (ijtsrd), ISSN: 2456-6470, Volume-9 | Issue-4, August 2025, pp.882-885, URL: www.ijtsrd.com/papers/ijtsrd97328.pdf

IJTSRD97328

I. INTRODUCTION

A diffuser is a duct where the flow decreases and the static pressure increases as the fluid moves from inlet to outlet. It's used widely in gas turbine engines, aircraft and jet engines for variety of purpose, such as supplying uniform flow in the head of combustor, or slowing the air delivered to the inlet face of the compressor or fan (Johnston 1998). Thus, study of the diffuser flow characteristic is important and method to predict the separation and attachment is one of the key challenges in flow simulation. Separation flows are difficult to predict because the separating and reattaching boundary layer are highly out of equilibrium. There are lots of experiments carried out to investigate the separation. Two-dimensional flow was investigated by Buice (2000) and Hoefener (2008). The velocity profile was obtained and the relationship with different inlet Reynolds number was presented. Cherry (2006) took the three-dimensional experiment to find out the characterization of the diffuser separation.

A lot of numerical studies carried out using Large-Eddy Simulation (LES)were conducted by many researchers. The different kinds of turbulence models were compared. Investigation into the predictive performance of linear and non-linear eddy-viscosity models and differential stress-transport closures for separated flow in a nominally two-dimensional asymmetric diffuser was done by Apsley (1999) and Leschziner (2006). It was demonstrated that advanced turbulence models using strain-dependent coefficients and anisotropy-resolving closure offer tangible advantages in predictive capability, although the quality of their performance can vary significantly, depending on the details of closure approximations adopted. But the second-moment models investigated aren't uniformly better than simpler closure strategies; no model can be said, without qualification and purely on the basis of agreement with experiment, to be clearly superior or inferior to others. Cherry (2006) compared the RANS, LES and LES-fine grid and the results presented that RANS simulation strongly over predict the strength of separation and the LES computations showed a much better agreement with the data, especially for the fine grid. LES of an incompressible planar, asymmetric diffuser flow was presented by Herbst (2007) with the dynamic smagorinsky model at different Reynolds numbers. It was found that the size of the main separation region is governed by inflow which penetrates further into diffuser at higher Reynolds number and increasing the Reynolds number a larger separated region is evident. Although there so many advanced models have been proposed to predict the separation, due to the limit of the computer resource, the common turbulence models are still widely used now.

Recently, there are many turbulent models developed but none of them is good enough (Wilcox 200I) and different problems required different closure models. In this paper, five common turbulence models are adopted in simulating the planar diffuser flow and the predictive abilities are compared and analyzed systematically in order to supply the foundation to choose turbulence models, modify the present models and develop the method modeling in multi-blocks.

II. MODELLING :

This 3d model is done in the CATIA V5 software with the commands of the Pad , Pocket , rib tool .

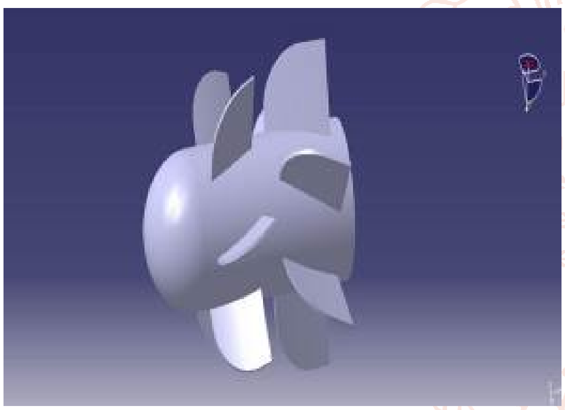

FIG 1. NORMAL BLADE

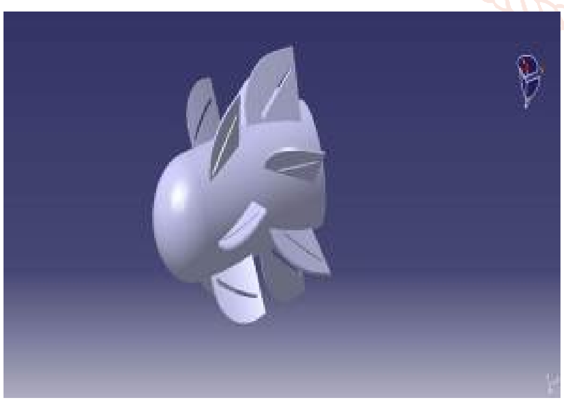

FIG2. MODIFIED BLADE

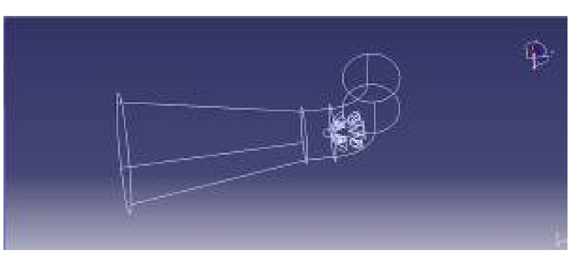

FIG 3. 3D MODEL OF KAPLAN TURBINE

MESH ANALYSIS :

Meshing of the Kaplan turbines made with the tetrahedrons elements with the nodes and elements of 20510 nodes and elements of 106214 for the modified blade models 19182 nodes and 98539 elements with high resolutions of mesh qualities

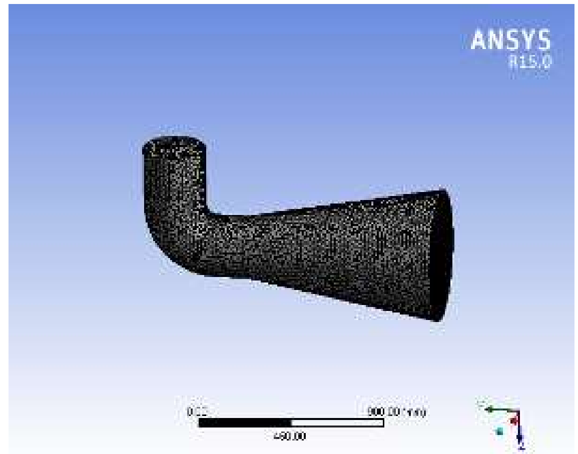

Fig: Mesh model of Kaplan Turbine

BOUNDARY CONDITIONS :

The boundary conditions for the CFD Analysis is pressure based solvers and SST K-omega Turbulence model is used the inlet flow velocity for this analysis 10m/s and blade rotational velocity of 1500 RPM, The CFD Analysis is iterations based 141 iterations taken for Normal blade analysis and 119 iterations taken for Modified analysis

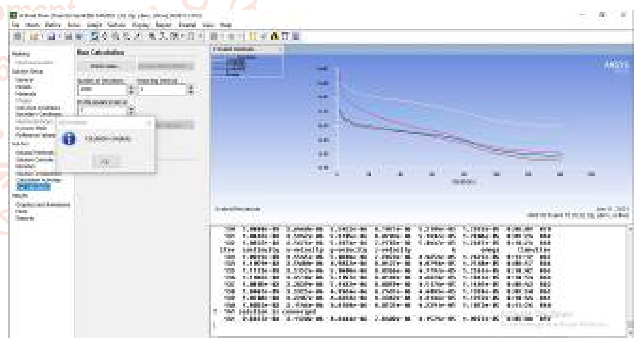

FIG 7. BOUNDARY CONDITIONS OF THE NORMAL BLADE

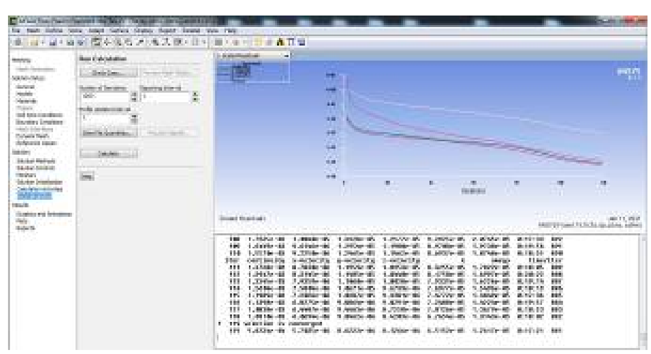

FIG 8. BOUNDARY CONDITIONS OF THE MODIFIED BLADE

III. RESULTS AND DISCUSSIONS :

NORMAL BLADE :

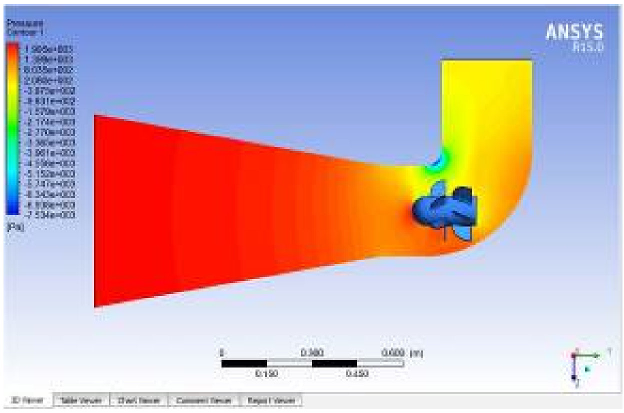

FIG 9 : PRESSURE OF NORMAL BLADE

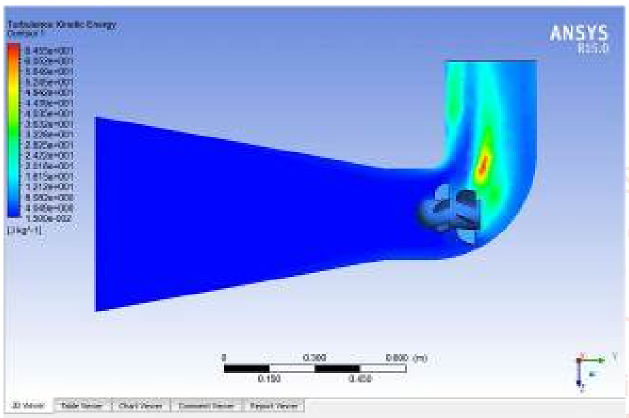

FIG 10 : TURBULENCE OF NORMAL BLADE

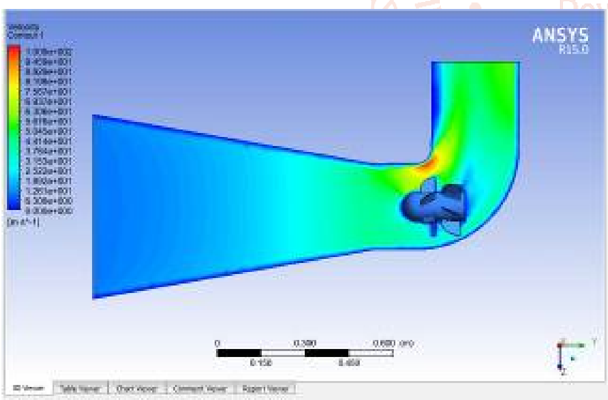

FIG 11 : VELOCITY OF NORMAL BLADE

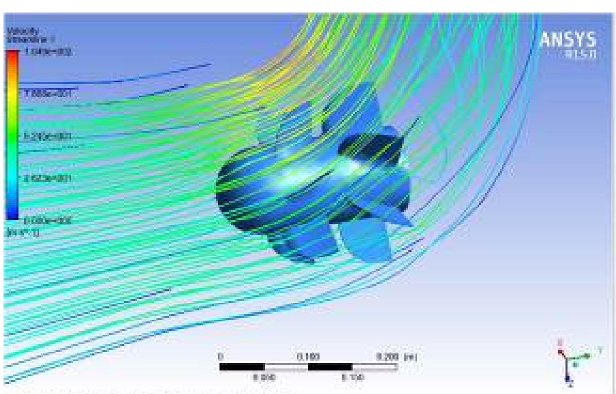

FIG 12 : VELOCITY STREAMLINE OF NORMAL BLADE

MODIFIED BLADE:

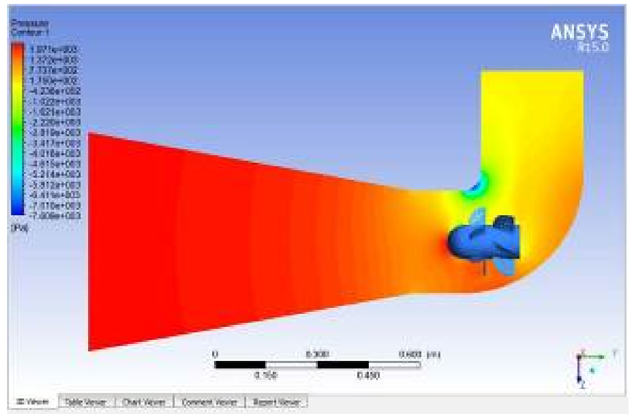

FIG 13 : PRESSURE OF NORMAL BLADE

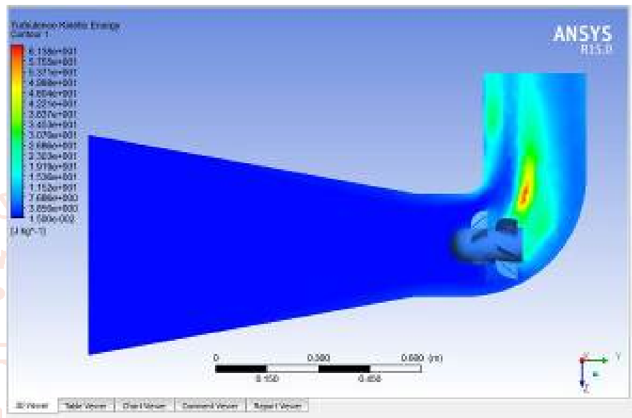

FIG 14 : TURBULENCE OF MODIFIED BLADE

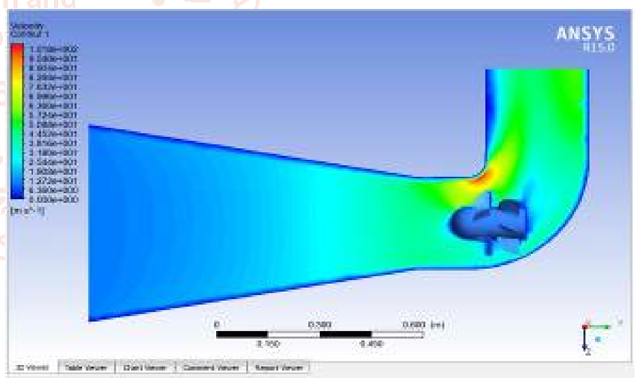

FIG 15: VELOCITY OF MODIFIED BLADE

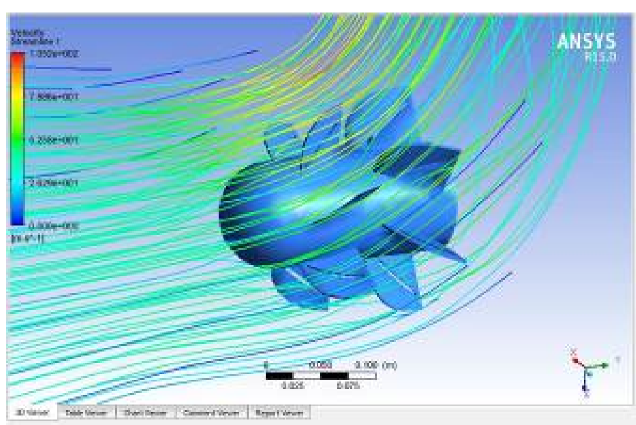

FIG 16: VELOCITY STREAMLINE OF MODIFIED BLADE

TABLE 1 . PRESSURE RESULTS

RESULTS	NORMAL BLADE	MODEIFIED BLADE
PRESSURE	2.00E+03	1.97E+03

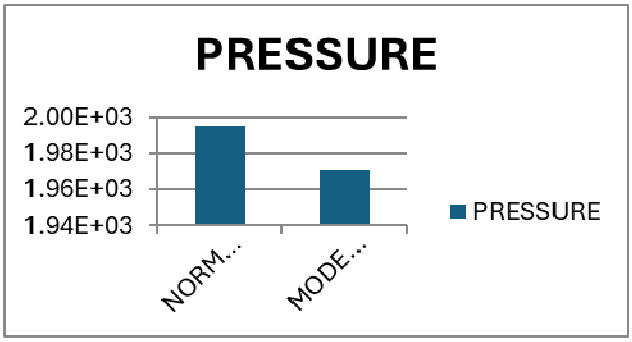

FIG 17 .PRESSURE RESULTS OF NORMAL AND MODIFIED BLADE

TABLE 2 . TURBULENCE RESULTS

RESULTS	NORMAL BLADE	MODEIFIED BLADE
TURBULENCE	6.46E+01	6.14E+01

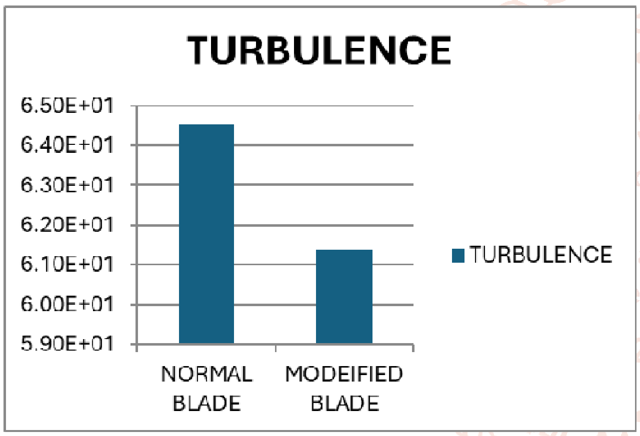

FIG 18 .TURBULENCE RESULTS OF NORMAL AND MODIFIED BLADE

TABLE 3 . VELOCITY RESULTS

RESULTS	NORMAL BLADE	MODEIFIED BLADE
VELOCITY	1.01E+02	1.02E+02

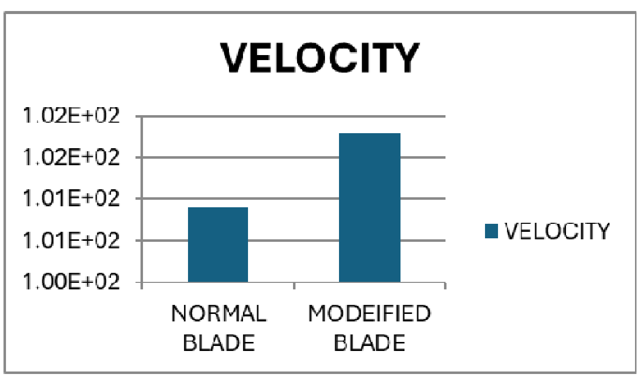

FIG 19 .VELOCITY RESULTS OF NORMAL AND MODIFIED BLADE

IV. CONCLUSION :

The CFD analysis of the Kaplan turbines done and flow performances can be validate through ANSYS Fluent Software the results of Pressure, Velocity and Turbulences can be analyzed for normal blade and Modified blades, the turbulence intensity is reduced in the modified blade and pressure drop can be reduced in the modified blades, the flow velocity can be increased in the modified blades, the streamline flow represents the flow pattern in the blade surface the modified blade get the smooth flow in the kaplan turbine.

REFERENCES :

[1] Apsley, D. D. and Leschziner, M. A., 1999, "Advanced turbulence modeling of separated flow in a diffuser", Flow", Turbulence and Combustion, Vo1.63, pp. 81 - 112.

[2] Buice, Carl. U. Eaton, John. K., 2000, "Experimental investigation of flow through an asymmetric planar diffuser", Journal of fluids engineering, Vo1.5, pp. 433 - 435.

[3] Cherry, E. M. and Glaccarino, 2006, "Separated flow in a three dimensional diffuser: preliminary validation", Center for Turbulence Research Annual Research Briefs.

[4] Durbin, P. A., 1995, "Separated flow computations with the model", AIAA Journal, Vol.33, pp. 659 - 664.

[5] Herbst A. H and Henningson D. S., 2007, "Simulations of turbulent flow in a planar asymmetric diffuser", Flow turbulence combustion, Vo1.79, pp. 275 - 306.

[6] Hoefener, L. Nitsche, W, 2008, "Experimental investigations of controlled transition in a laminar separation bubble at an axisymmetric diffuser", New results in numerical and experimental fluid mechanics VI, Vo1.96, pp. 244 - 251.

[7] Johnston, J. P., 1998, "Review: Diffuser Design and performance analysis by a unified integral method", Journal of fluids engineering, Vol. 3, pp. 6 - 18.

[8] Leschziner, M. A and Wang, C. et al., 2006, "Contribution by ICSTM: modeling generic 2D and 3D separated flow using anisotropy-resoling turbulence closures", Flomania-an European initiative onflow physics modeling, Vo1.94, pp. 77 - 84.