Design And Optimization Of Reaction Turbine Blade Through CFD Analysis

Hitesh G. Patel [1], Prof. A.N. Shyani [2]
[1, 2] Dept of Mechanical Engineering
[1] PG Student SAL Institute of Technology & Engineering Research, Ahmedabad, gujarat
[2] HOD, SAL College Of Engineering Ahmedabad, gujarat

Abstract- *CFD is the art of numerically solving the governing fluid flow equations in order to obtain the descriptions of the complete flow-field of interest. With the availability of computer power and efficient numerical algorithm, CFD is becoming a very important tool for engineers in improving the performance of component involving fluid flows. The reason is that CFD effectively replace the needs to perform expensive experimental measurement and testing of new design and prototype.*
Reaction type turbines (Francis) are commonly used in hydropower generation. Main components of the turbine are spiral case, stay vanes, guide vanes, turbine runner and the draft tube. The dimensions of these parts are dependent mainly on the
design discharge, head and the speed of the rotor of the generators. In this study, a methodology is developed for parametric optimization by Computational Fluid Dynamics (CFD) into the design process.
The performance of turbine blade is related to many factors. One of the important factor is the turbine blade angle. Blade angle of turbine mainly affect the turbine efficiency. The performance of turbine blade can be improved by using Computational Fluid
Dynamics (CFD).

Keywords- reaction turbine blade, Computational Fluid Dynamics, Francis turbine, Design, Optimization, Turbine blade, Modeling And Analysis.

I. INTRODUCTION

The flow of water is pressurized fully In reaction type turbines. The potential energy of water is again converted in to kinetic energy by a rise in speed. In reaction turbine runners, the energy is transferred to the runner by an action throughout the blades and not by a local impact and It uses the action- reaction principle. The water releases its energy continuously, which appears with a pressure decrease along the blades. The mostly used reaction turbines are Francis turbine and Kaplan turbine types.

II. DESCRIPTION OF THE PROJECT

In this project we are showing various effect on head and discharge pressure of francis turbine by changing the blade angle of francis turbine runner.we are changing the blade angle by using trial and error method. Number of readings are taken for this project work but here we have include only first three blade angles through which we got highest head and discharge pressure. And we get the optimized blade angle.

We have selected following turbine specification for further analysis.[9]

Model selected	F280
Desired P Generator output	3000 KW
Rated Head Available	48 m
Desired P turbine Output	3142 KW
Rated Flow	7.25 m^3/s
Rated N of turbine	600 rpm
Prototype runner diameter	1.01 m
Model runner diameter	0.35 m
Scale up ratio	2.88
Runner diameter	0.35 m
No. of runner blades	13
No. of guide vanes	18
PCD of Guide vanes	0.40 m
No. of stay vanes	18
Blade angle	18°

From the research paper we have taken the following experimental data and make a CFD analysis and make comparison between experimental data and CFD result.[9]

Table 1 CFD Experimental data

Sr No.	N11	P11	η	H (m)	Pr. (Pa)	P (KW)	Q	Q11
1	70	9	89	74.95	735221	5956.75	9.1	1.03
2	80	9.3	92.8	57.38	562904	4123.57	7.89	1.02
3	87.5	9.28	93.1	47.97	470542	3144.74	7.18	1.01
4	90	9.22	93	45.34	444764	2864.98	6.93	1.01
5	100	8.7	89.5	36.72	360259	1975.06	6.13	0.99

III. GEOMETRIC MODELLING, BOUNDRY CONDITION & COMPUTATIONAL PARAMETERS

First of all cad model is generated and after that CAD model is imported to ANSYS for further analysis

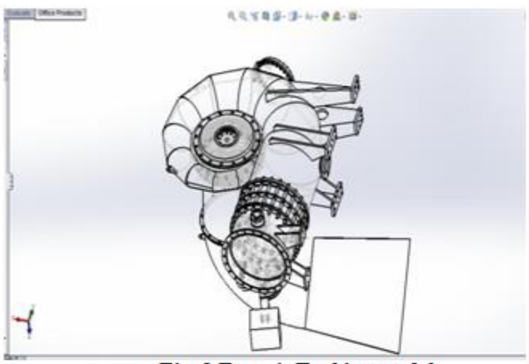

Fig. 1 Francis Turbine model

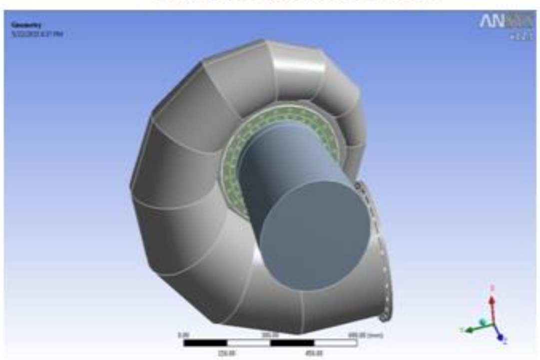

Fig. 2 3D cavity model of Turbine

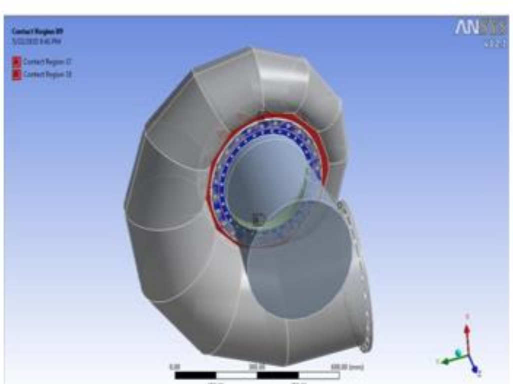

Fig. 3 Defining the constrain for turbine.

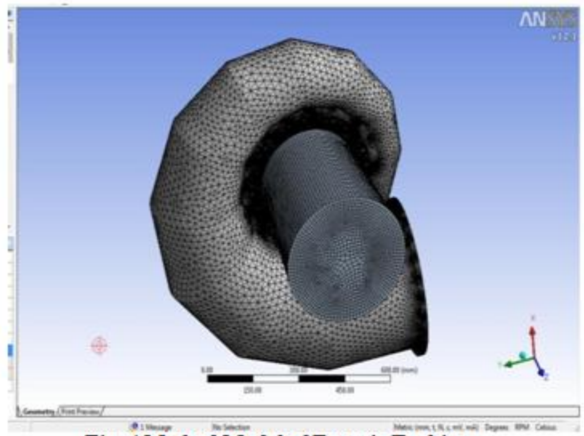

Fig. 4 Meshed Model of Francis Turbine

Table 2 Mesh characteristics of each turbine component for final design

Domain Part	No. of nodes	No. of elements	Type of elements
Casing (Including Stay vanes and Guide vanes	**204447**	**1605723**	**Tetrahedron**
Runner	**98741**	**382088**	**Tetrahedron**
Draft tube	**121845**	**35961**	**Tetrahedron**

After Generation of mesh on model the model is imported in ANSYS CFX.

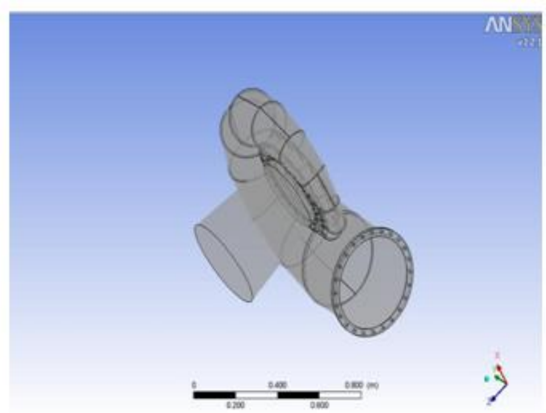

Fig. 5 Francis Turbine in ANSYS CFX

In ANSYS CFX the domain is defined for casing. The details of type of domain used , domain fluid , and the motion of domain is shown below.

Domain Type:- Fluid Domain

Domain Fluid:- Water

Domain Motion:- Stationary

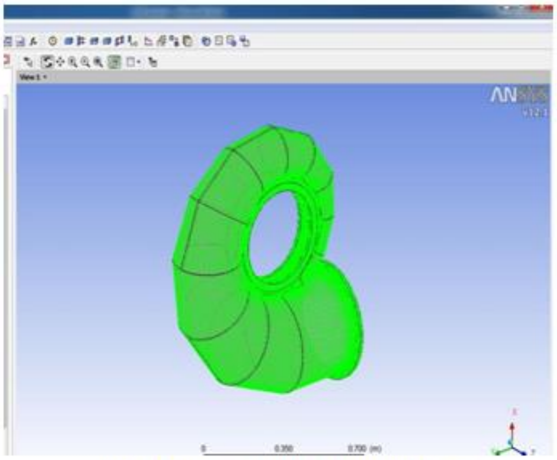

Fig. 6 Stationary Water Domain for casing

Once domain is defined the next step is define boundry condition. Here we have defined two boundary condition.

1) Outlet Boundary Condition

2) Inlet Boundary Condition

In Outlet Boundary Condition Mass Flow Remain Constant and in inlet boundary condition mass flow rate is taken as 7.25 m^3/s.

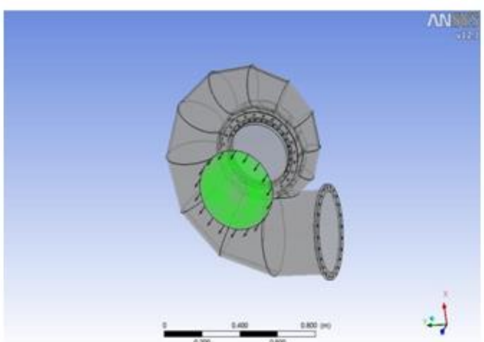

Fig. 7 Defining the Outlet Boundary Condition

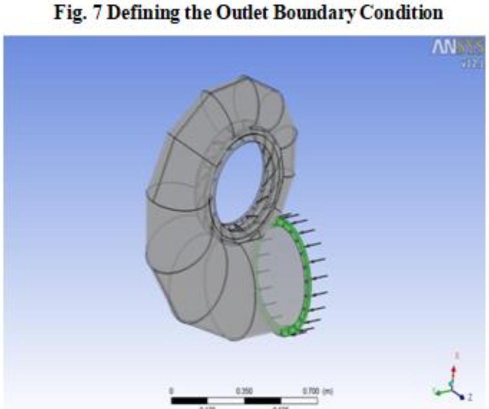

Fig. 8 Defining the Inlet Boundary Condition

After defining boundry condition pre processer is running in the CFX software and we get the analysis result. Table of this result is as shown below.

Table 3.CFD Analysis Results

Sr No.	N11	H	Pr.
1	70	69.82	735235
2	80	56.57	562965
3	87.5	45.82	471846
4	90	43.97	468157
5	100	34.28	372158

Now, By comparing the Experimental results and CFD analysis result a graph is generated by the software which is shown as below. A graph clearly shows that there is no much difference between the experimental result and CFD Result.

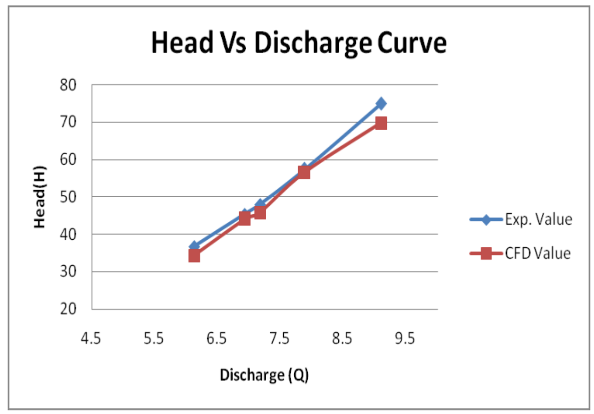

Fig. 9 comparison experimental result & CFD result

After making comparison we have validate our model and now we are going to make analysis on various effects of blade angle on model. We first make analysis at the model's blade angle at 18 degree and then head and pressure is notice. After doing that we make a change in blade angle and note down the results.

IV. EXPERIMENTAL INVESTIGATION

Pressure Contour of Turbine Casing at blade angle 18 degree

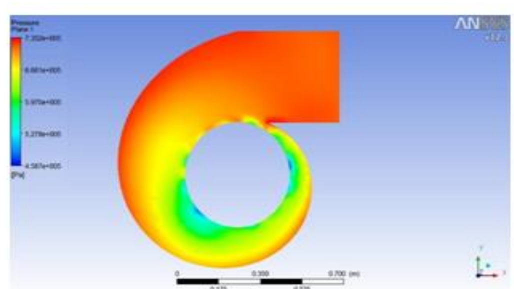

Fig. 10 Pressure Contour of Turbine casing at actual blade angle (i.e 18 degree)

Table 4 Head and pressure at blade angle

Sr No.	N11	Blade angle	H (m)	Pr.(Pa)
1	70	18°	69.03	735281

As we can see that from above table at the blade angle we get head in m 69.03 and pressure in Pascal 735281 , which is as shown in above figure.

Now for making optimization number of blade angles are changed and analysis is performed on model using Computational Fluid Dynamic (CFD).But in this thesis we have only show three highest blade angles which have maximum head and pressure.

Cases for changing blade angle:

1] First Case –

Optimization using Runner Blade Angle Increase by 2 Degree

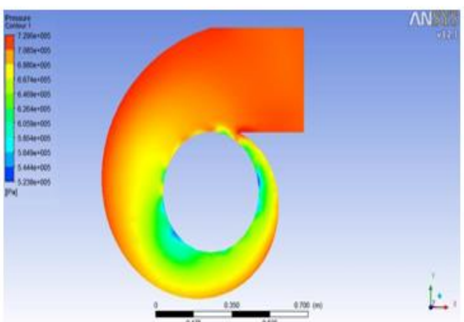

Fig. 11 Pressure Contour of Turbine casing at blade angle (i.e 20 degree)

Table 5. Head and pressure at blade angle 20

Sr. No.	N11	Blade angle	H (m)	Pr.(Pa)
1	70	20°	68.07	728987

As we can see that from above table at the blade angle 20 degree we get head in m 68.07 and pressure in Pascal 728987 which is as shown in above figure. But at this blade angle we get head and pressure lower than at blade angle 18 degree. So we have decided for decrease the blade angle.

[2] Second Case

Optimization using Runner Blade Angle decrease by 2 Degree

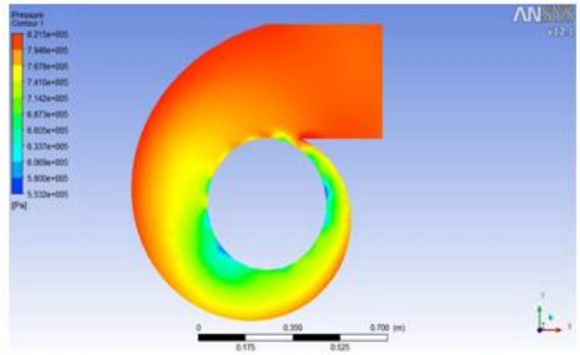

Fig. 12 Pressure Contour of Turbine casing at blade angle (i.e 16 degree)

Table 6 Head and pressure at blade angle16

Sr No.	**N11**	**Blade angle**	**H (m)**	**Pr.(Pa)**
1	**70**	16°	**70.89**	**821458**

As we can see that from above table at the blade angle 16 degree we get head in m 70.89 and pressure in Pascal 821458 which is as shown in above figure.

At this blade angle we get head and pressure higher than at blade angle18 degree. So we have decided for decrease the blade angle again.

[3] Third Case

Optimization using Runner Blade Angle decrease by 3 Degree

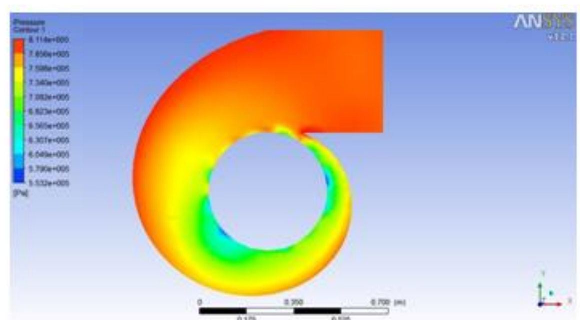

Fig. 13 Pressure Contour of Turbine casing at blade angle (i.e 15 degree)

Table 7. Head and pressure at blade angle 15

Sr. No.	N11	Blade angle	H (m)	Pr. (Pa)
1	70	15°	69.28	821431

As we can see that from above table at the blade angle 15 degree we get head in m 69.28 and pressure in Pascal 821431 which is as shown in above figure.

At this blade angle we get head and pressure lower than at blade angle 16 degree.

Now we have compared these three results and make selection for optimized blade angle. Comparison table of these three blade angle is as below.

Table 8.Comparison between four cases of blade angle

Sr. No.	N11	Blade angle	H (m)	Pr. (Pa)
1	70	18	69.03	735281
2	70	20	68.07	728987
3	70	16	70.89	821458
4	70	15	69.28	821431

V. RESULTS AND DISCUSSION

We can note from above table is that at blade angle 16 degree we get the highest head and highest pressure so that we conclude that 16 degree blade angle is optimized blade angle. Finally from above conclusion we have choose 16 degree blade angle as optimized blade angle and at designing level of runner blade we choose 16 degree blade angle and make a final blade design for getting maximum head and pressure.

Final Blade Design

Final blade shape is obtained from above simulation. It is possible to say that flow separation does not occur, because the meridional vectors follow the meridional paths as plotted in Fig. 14. and the pressure distribution supports this observation as shown in Fig 15

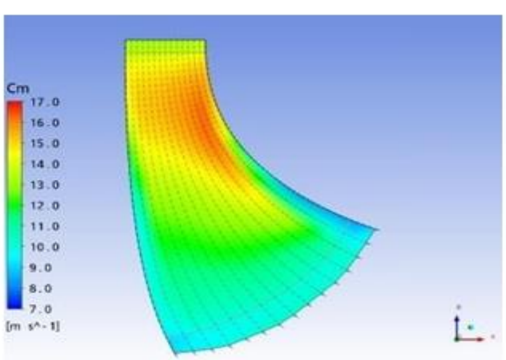

Fig. 14 Meridional flow velocity vectors (i.e 16 degree)

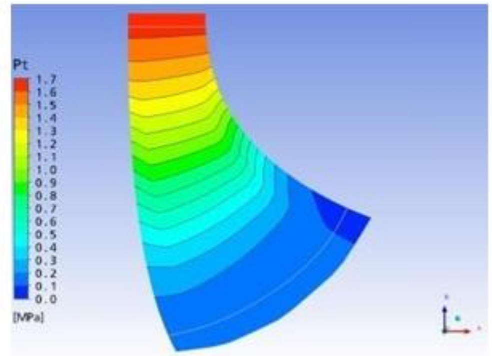

Fig. 15 Variation of total pressure on meridional section (i.e 16 degree)

The velocity vectors shown in Fig.16 follow the blade profile throughout the runner passage. Any flow separation on the blade is not expected in the meridional section from inlet to outlet.

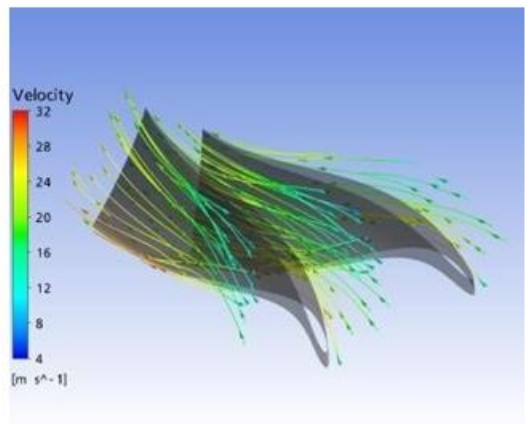

Fig. 16 Velocity streamlines (i.e 16 degree)

From above figure it is shown that the velocity stream line are continuously increasing in value as they are move toward the centre from the blade outer edge, Also the initial Velocity of the stream line at the time of entrance is approx 7 m/s but as the fluid flows inside it follow the blade profile so that there is continuously pressure is decrease and it converted in to velocity so that near to hub we achieve maximum velocity.

VI. CONCLUSION

The paper brought out the validation of experimental results with computational investigation. The model is first validated by comparison between experimental results and CFD analysis result . Then by using trial and error method we have make a changes in various blade angles.

Blades acts like a heart in any type of turbine, as they are the principal elements. The efficiency and reliability are two parameters of a turbine which depend on the proper design of the blades.

By using trial and error method we have choose various blade angle. Number of blade angle changed and results are taken but we have discussed on only three results with highest head and discharge pressure.

By changing blade angle we found that at 16 degree we get highest head and highest discharge pressure. Finalize blade design is prepared at 16 degree blade angle.

Prediction of turbine performance by CFD gives the idea to know the flow behavior inside the turbine model and get the information about intricacy of flow pattern, since the flow behavior inside the turbine in actual is very complicated. CFD result gives the qualitative information. It provides the tool to simulate the flow condition with different geometries in lowest possible time, thus providing reduction in design analysis and yet developing roboust technology along with aiding in reducing gestation period.

NOMENCLATURE

H	Net Head
Q	Discharge Through Turbine
N	Rotational Speed Of The Turbine
g	Gravitional Acceleration
P	Turbine Power in KW
PCD	Pitch Circle Diameter
N_{11}	Unit Speed
Q_{11}	Unit Discharge
P_{11}	Unit Power
Pr.	Pressure (Pa)
Q_d	Design Discharge
H_d	Design head
F	Frequency in Hertz
ω	Rotational speed of turbine

REFERENCES

[1] A Textbook of Fluid Mechanics by R.K. Rajput

[2] R.A. Saeed , A.N. Galybin, V. Popov (May – 2013) "Modelling of flow-induced stressesin a Francis turbine runner." 2010 Elsevier Ltd.

[3] Xavier Escaler, Eduard Egusquiza, Mohamed Farhat, Franc-ois Avellan,Miguel Coussirat (2006) "Detection of cavitation in hydraulic turbines" Elsevier Ltd.

[4] C.G. Rodriguez, E. Egusquiza,_, X. Escaler, Q.W. Liang, F. Avellan (2006) "Experimental investigation of added mass effects on a Francis turbine runner in still water" Elsevier Ltd.

[5] WANG Fu-jun, LI Xiao-qin, MA Jia-mei, YANG Min (2009) "Experimental investigation of characteristic frequency in unstudy hydraulic behaviour of a large hydraulic turbine" Elsevier Ltd.

[6] R.A. Saeed *, A.N. Galybin (2009) "Simplified model of the turbine runner blade" Elsevier Ltd.

[7] SUSAN-RESIGA , Gabriel Dan (2006)"Jet Control of the Draft Tube Vortex Rope in Francis Turbines at Partial Discharge"

[8] J D Denton and W N Dawes "Computational fluid dynamics for turbomachiner design"

[9] Manoj kumar shukla, Rajeev jain, Prof. Vishnu Prasad, S.N. Shukla. " CFD analysis of 3-D flow for francis turbine." MIT Publications.