“FLOW ANALYSIS OF FRANCIS TURBINE”

PROJECT MEMBERS : **GROUP ID** : 130009449

SISODIYA DIGVIJAY M. (10ME33)

PATEL HARDIK R. (10ME14)

NAYI CHANDRESH A. (10ME03)

PATEL HARSH P. (10ME16)

INTERNAL GUIDE : MR.JAYENDRA B. PATEL

Department of Mechanical engineering,
Smt. S. R. Patel Engineering College,
Dabhi,Unjha-384170

Gujarat Technical University

न हि ज्ञानेन सदृशं पवित्रमिह विद्यते।

Smt. S. R. PATEL ENGINEERING COLLEGE

REASON FOR SELECTING OF PROJECT ?

- Turbine is used for converting first Hydraulic energy into Mechanical energy and then into Electrical energy . This energy may used in Domestic and Industrial applications.

- If any fault in the parts of turbine then proper conversion of energy may not be possible . Means we can’t get desired output . So, that we put machine in the maintenance section , so that cost of it may add. Therefore , loss of money as well as living lives.

- From the analysis , we will find Location of various Stresses and Pressure distribution in the turbine and try add some features in the design of turbine to reduce or eliminate problem at possible extent.

PROJECT BACKGROUND :

- The word turbine was introduced by the French engineer CLAUDE BURDIN in early 19th century. Waterwheels have been used for thousands of years for industrial power. The main thing in turbine are HEAD and FLOW RATE.

- JAN ANDREJ SEGNER developed a reactive water turbine (Segner wheel) in the mid 18th century. It had a horizontal axis. Segner works with Euler on some early mathematical theories of turbine design.

- In 1820, JEAN-VICTOR PONCELET developed an inward flow turbine.

- In 1826, BENOIT FOURNEYRON developed an outward flow turbine which is having efficiency around 80%.

- In 1844, URLAH A. BOYDEN improved on performance of fourneyron turbine.

- In 1849, Sir JAMES B. FRANCIS improved the inward flow reaction turbine to over 90% efficiency. The Francis turbine is the 1^{st} modern water turbine. It is also a radial flow turbine.

- Around 1890, the modern fluid bearing was invented, is used to support heavy water turbine spindles.

- Around 1913, VIKTOR KAPLAN created the kaplan turbine. It was evolution of Francis turbine but revolutionized the ability to develop low-head hydro sites.

- The Francis design was subsequently improved but changing the shape of the runner blades so that the water was turned from a radial to an axial path within the runner rather than outside it. During all the changes made of this type of turbine, it was the name of Francis that become and remained associated with it.

OBJECTIVE :

- To innovate optimization on Francis Turbine design.
- To design and develop new Francis turbine.
- To improve runner & casing design of Francis turbine and analyze it's performance with the CFD.
- To increase the efficiency of Francis turbine.
- To decrease the frictional losses, runner losses and draft tube losses.
- To reduce the vibration of the Francis turbine.

METHODOLOGY :

Find the definition of project

↓

Read the research paper

↓

Dismantle the model

↓

Take dimensions of all parts of model

↓

Generating the fluid model of casing & runner

↓

Based on research paper apply the boundary condition in CFD analysis of model

↓

Calculate pressure counter, velocity counter &flow analysis of model

↓

Result analysis & discussion

↓

Future scope & conclusion

SCOPE OF PROJECT :

- The results can be used to optimize the runner and to analyze other critical components in the hydraulic turbine.
- Put the different pressure counter & velocity counter in CFD and improve the design of francis turbine.
- The analysis of "problem due to erosion" is apply at the location where this problem may present . So ,eliminated or reduced that problem.
- For reduction in Stress Concentration like, in runner of turbine.

LITERATURE REVIEW :

Sanjay Jain[1] :

CFD approach for prediction of efficiency of francis turbine

Boundary conditions : (i)Pressure inlet and Pressure outlet , and
(ii)Mass flow inlet and outlet

- ❑ CFD approach complements the other approaches, as CFD approach helps in reduction in cost of model testing and saving in time which leads to cost effective design of the system.

- ❑ CFD approach may be helpful in improvement of the existing efficiency measuring techniques and evaluation of the performance of hydro turbines to enhance the viability of hydropower development.

Krishna Prasad[2]:

Innovative Design of Francis Turbine

Aim: Elimination of sand erosion problem.

Use of CFD tool :

The pressure and velocity of the fluid is determined along the pressure side and suction side of the blade design.

- Erosion problem cannot stop completely by current technology but it can be minimize economically acceptable way by incorporating new design approached.
- Technological challenges and opportunities that recognize due to innovative design of Francis turbine should adopt by the turbine manufacturers.

- **<u>R.A. Saeed[3]:</u>**

"Modelling of flow-induced stresses in a Francis turbine runner "

Water flow that has been investigated by using Computational Fluid Dynamics (CFD) in order to identify the loads acting on the turbine blades.
The finite element analysis of stresses has been performed based on the pressure distributions calculated from CFD modelling.

Stress analysis:

In this study the analysis has been performed by using ANSYS that an effective tool for modelling stresses in Francis turbine runners.

- Velocity and pressure are highly sensitive to the operation conditions. It has been found that the stresses in the trailing edge of the runner blade near the crown reach a critical state in all operating points.

- **R.A. Saeed[4]:**

“Simplified model of the turbine runner blade ”

The main objective :To determine the maximum stress in the simplified blade model for the entire period of operation.

Stress analysis:

Stresses induced by water pressure in a whole Francis turbine runner have been calculated by using FEM for some loading conditions . The load caused by water pressure was derived from the CFD .

- The real data on different operation conditions .

➢ <u>XIAO Ruofu[5]:</u>

“Dynamic Stresses in a Francis Turbine Runner Based on Fluid-Structure Interaction Analysis ”

Computational fluid dynamics (CFD):

To analyze the pressure distribution on the runner Surface.

Fluid-structure interactions (FSI): To calculate the stresses in the Francis turbine runner to analyze the static stress characteristics in the Francis turbine runner at various operating points.

Operating Points : Total 11 points, which including the highest head 79m, and the lowest head 51 m.

- ❑ To get more accurate result we should put more Operating Points during analysis .

- **Z. Carija and Z. Mrsa[6]:**

" Complete Francis turbine flow simulation for the whole range of discharges"

This is a steady-state approximation where individual cell zones move at different rotational speeds. This approach is appropriate when the flow at the boundary between these zones is nearly uniform.

- The lowest pressures are on the suction side near the trailing edge. This is the position where cavitation occurs at the high loads.

- *The main idea is to assume steady state uniform conditions in circumferential direction at the spiral-casing outflow.*

➢ **<u>Ravindra R.[7]:</u>**

"CFD Analysis Of Francis Turbine"

The aim of this paper is to analyse the turbine for mechanical failure along with predicting its performance under actual operating condition by using CFD which is generally constrained by using a prototype.

The experimental approach of evaluating the performance of Francis turbine is costly as well as time consuming. Conversely CFD approach is faster and large amount of results can be produced at virtually no added cost.

The CFD approach for the prediction of efficiency of Francis turbine was developed with accomplishment of analysis of Francis Turbine performance.

From this we find only importance of Use Of CFD.

➢**<u>RADU NEGRU[8]:</u>**

"*Analysis Of Flow Induced Stress Field In A Francis Turbine Runner Blade*"

The geometrical model was reduced to **<u>one blade</u>**, due to the periodical symmetry of the runner. The pressure field obtained from computational fluid dynamics (CFD) was applied as a mechanical load on the blade surface in the structural finite element analysis (FEA).

In order to obtain the stress distribution on the blade , the loads due to the water pressure, to the centrifugal force induced by rotation and to the own weight were considered.

The maximum stresses occur at the transition between the blade and the crown, in the trailing edge area, with a rapid decrease toward the transition to band. For the leading edge, the highest stresses occur at the transition to band.

➢ **<u>Mohammed Asid Zullah[10]:</u>**

"CFD validation of performance improvement of a 500 kW Francis turbine"

A CFD-based design system , which integrates a full Francis turbine is presented in this paper. CFD allows a quick and efficient improvement and optimization of turbine components.

The highly successful combination of the CFD-based design optimization with model testing has finally resulted in a new model which can provide about 9.93% upgrade in peak efficiency. CFD approach may be helpful in improvement of the existing efficiency measuring techniques.

- ❑ CFD helps to find out Improvement in working condition by increment in efficiency.

- **Yongzhong Zeng[11]:**

"Prediction and experimental verification of vortex flow in draft tube of Francis turbine based on CFD"

Starting from computation fluid dynamics, to calculate the *unsteady flow* in the turbine was analysed in this paper. The geometric physical model of the whole flow passage of a Francis turbine was established. And FLUENT software was used to numerically simulate the flow in the turbine.

These researches play a decisive role in the optimal design of turbine, the predication of stable operating zone of the unit, the guaranteeing of stable operation of the unit, and the improvement of power quality.

SPECIFICATION OF FRANCIS TURBINE :

1. Net Head : 20 meters (Approximately)
2. Discharge : 2000 LPM
3. Speed : 1500 RPM
4. Motor and Pump : 15 HP

- The input into the turbine is controlled by a set of guide vanes. The net supply head is measured by means of a differential manometer. For the measurement of speed, Tachometer is to be used. A straight conical draught tube is provided vertically after the runner. Rope brake arrangement with suitable pulleys is provided for loading the turbine.

STANDARD DATA :

g = Acceleration due to gravity = 9.81 m/s^2
ρ_w = Density of water = 1000 kg/m^3
ρ_m = Density of manometer = 13550 kg/m^3
D = Diameter of brake drum = 0.35 m
C_d = Co-efficient of discharge = 0.69

Venturimeter :

- Diameter of converging = 0.065m
- Area of pipe inlet A_1 = 0.00331 m^2
- Diverging section = 0.065 m
- Diameter of throat = 0.039 m
- Area of throat A_2 = 0.00119 m^2
- Co-efficient of discharge C_d = 0.69
- Area Ratio A_1/A_2 = 0.0000039487

FORMULA USED :

❑ Total head :-

$H = 10(P_d + P_s/760)$ m of H_2O

P_d = Pressure Gauge reading

P_s = Vacuum of HG

❑ Velocity :-

$v = C_d \sqrt{\{(2gh/100)(\rho_m/\rho_w - 1)\}}$ m/s

$v = C_d \sqrt{\{(2gh/100)(12.6)\}}$

h = Manometric difference in cm

ρ_m = Density of manometer fluid i.e. Hg = 13550 kg/m^3

ρ_w = Density of water = 1000 kg/m^3

g = 9.81 m/s^2

❑ Discharge :-

$$Q = CA_2\sqrt{[2g(h_1-h_2)]/[1-(A_2-A_1)^2]}$$

❑ H.P. Hydraulic (Input) = vgHQ kW

❑ B.H.P. (Output) :-

B.H.P. = $[(2\pi NT)/60000]$ kW

$T = (W_1-W_2)*D*g$

Rope with hanger weight = 1.0

Where,

W_1= weight + hanger with rope reading

W_2= spring balance

D= diameter of the brake drum

N= number of R.P.M. of brake drum

CALCULATION TABLE :

Sr. no.	RPM N	Pressure Gauge Reading P	Diff. pressure of Manometer reading			Velo-city m/s V	Q= AV	I/P kW	O/P kW	Efficienc-y
			h 1	h 2	h= (h1-h2)					
1.	1258	1.6	57.8	14.2	43.6	6.64	.033	5.26	3.25	61.68
2.	1300	1.65	59.7	14.6	45.03	6.86	.034	5.42	3.35	61.80
3.	1364	1.73	62.6	15.4	47.2	7.18	.036	5.73	3.54	61.78
4.	1412	1.8	64.9	16	48.9	7.44	.037	5.9	3.64	61.69
5.	1456	1.85	66.8	16.4	50.4	7.67	.038	6.05	3.74	61.81

DISMANTLE THE TURBINE & TAKE DIMENSIONS :

Here, our project is totally follow reverse engineering of hydraulic machine(Francis Turbine).

In Reverse Engineering we follow steps like –i.) Dismantled , ii.) Take Dimensions ,then iii.) Design , once all these steps follow perfectly ,at the end we can done analysis on basis of that. In which we required all important dimensions of working model of a turbine for that we dismantled the machine and take all dimensions manually by necessary instruments.

For analysis purpose , without nearer to exact dimensions we can't analyzed model efficiently so finally our aim may not satisfy . That is loss of everything.

BASIC INFORMATION ABOUT CFD :

What is Fluid Flow ?

- Meteorological phenomena (rain, wind, hurricanes, floods, fires)
- Environmental hazards (air pollution, transport of contaminants)
- Complex flows in furnaces, heat exchangers, chemical reactors et

What is CFD ?

Computational fluid dynamics (CFD) provides a qualitative (and sometimes even quantitative) prediction of fluid flows by means of :

- Mathematical modelling (partial differential equations)
- Numerical methods (discretization and solution techniques)
- Software tools (solvers, pre and post processing utilities)

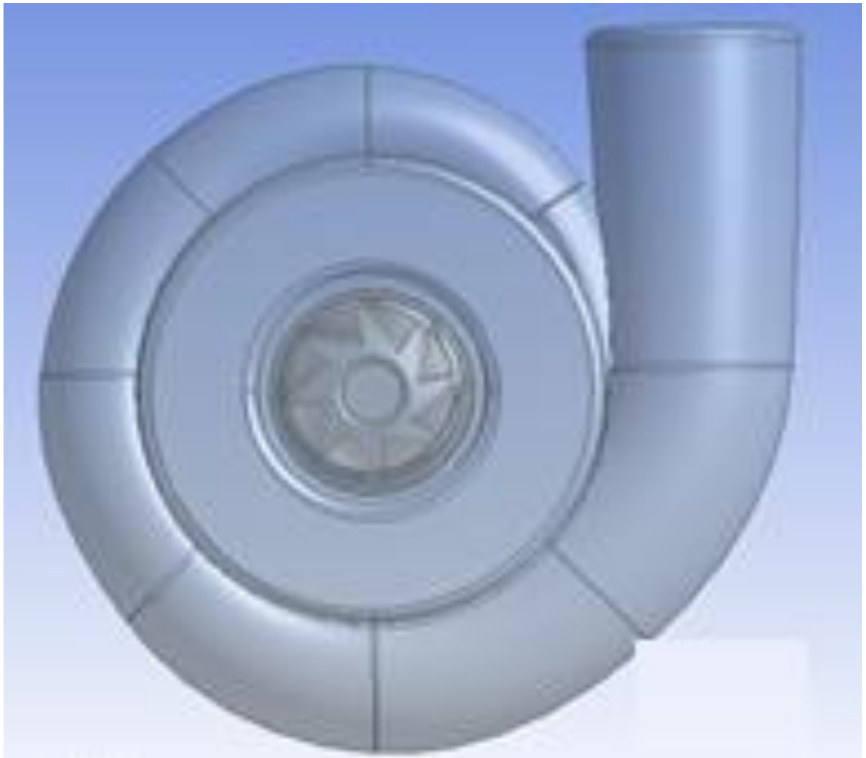
GENERATION OF FLUID MODEL :

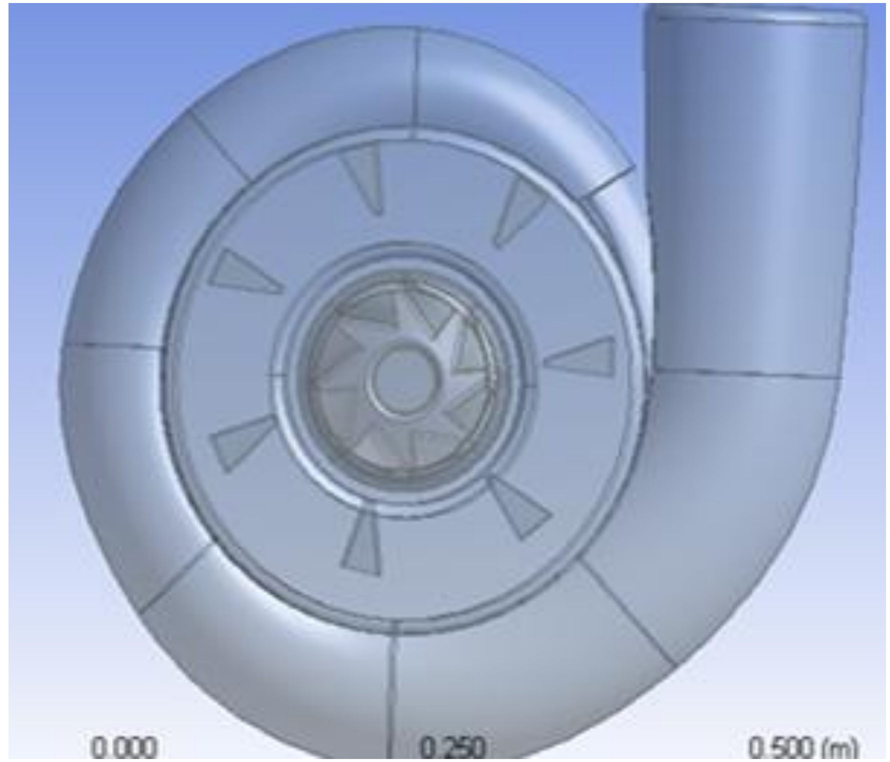
- Fluid model is considered for numerical solution. Fluid model is having different topology compared to actual model. In fluid model area of interest for numerical study is only working domain of fluid.

- Fluid models are generated in ANSYS R15.0 .

GENERATION OF FLUID MODEL :

WITHOUT DRAFT TUBE :

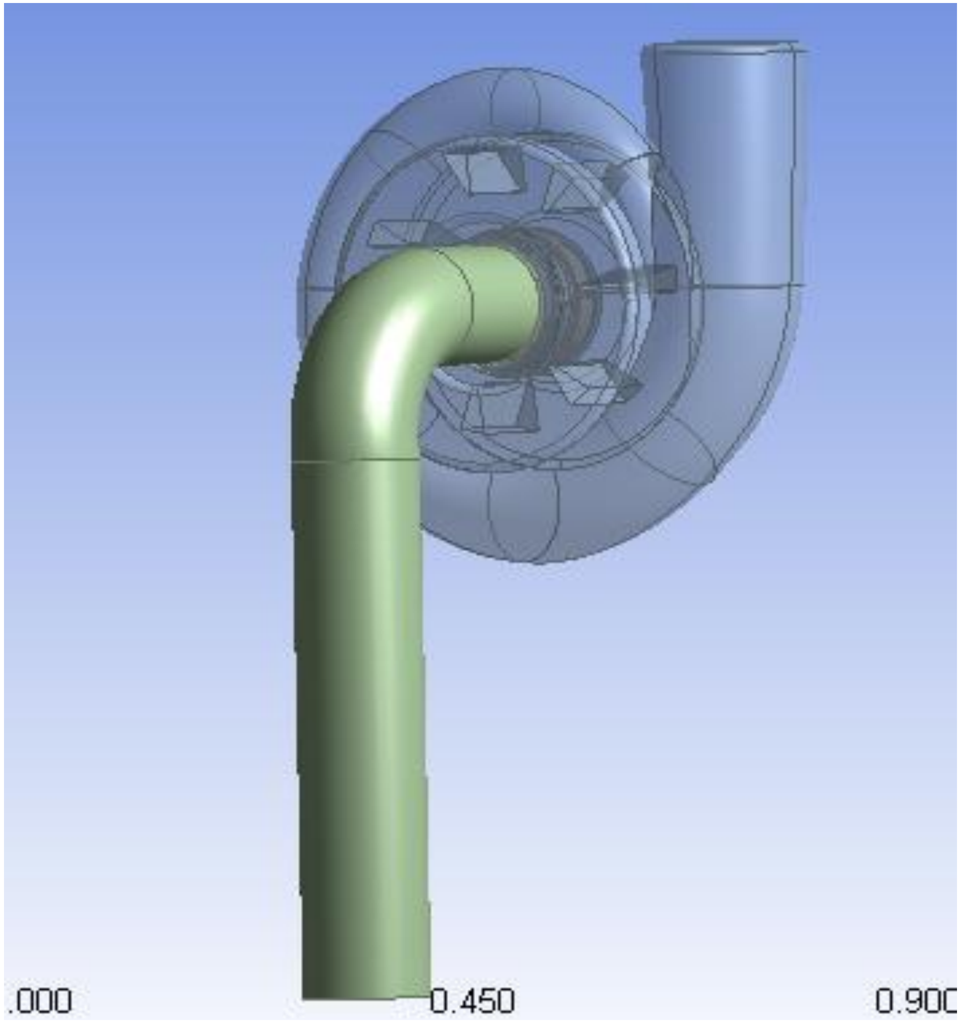




WITHOUT GUIDE VANE

WITH GUIDE VANE

GENERATION OF FLUID MODEL :



WITH GUIDE VANE AND DRAFT TUBE :

GENERATION OF MESHING :

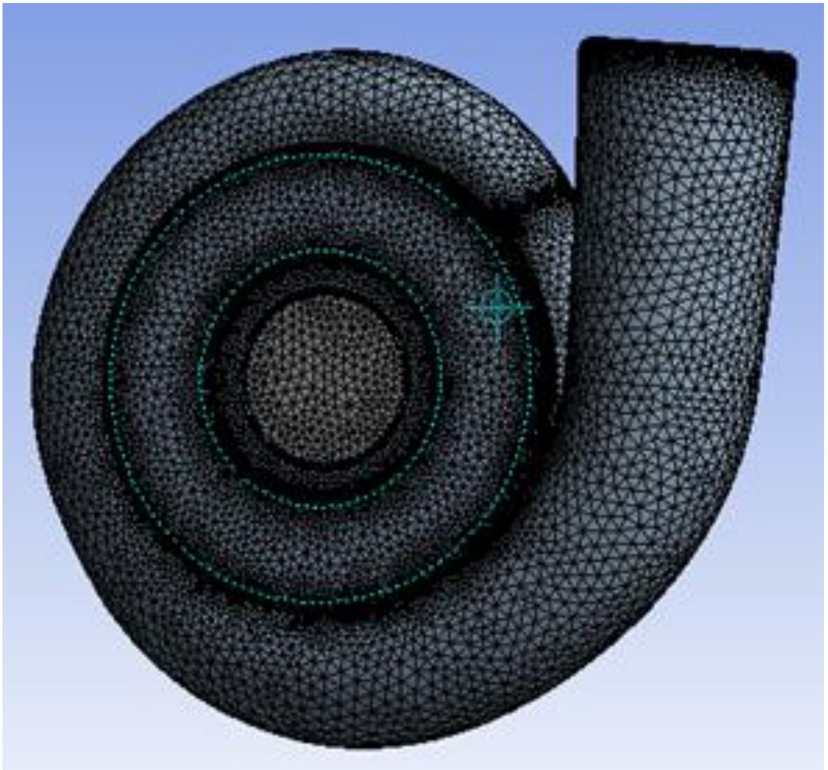
- In the numerical solution the working domain is divided into small sub domains. These sub domains are called as mesh. Governing equations are then descretised and solved in each of these sub domains. There are two types of mesh; structured and unstructured.

- A structured mesh is characterized by regular connectivity that can be expressed as a two or three dimensional array. This restricts the element choices to quadrilaterals in 2D or hexahedra in 3D.

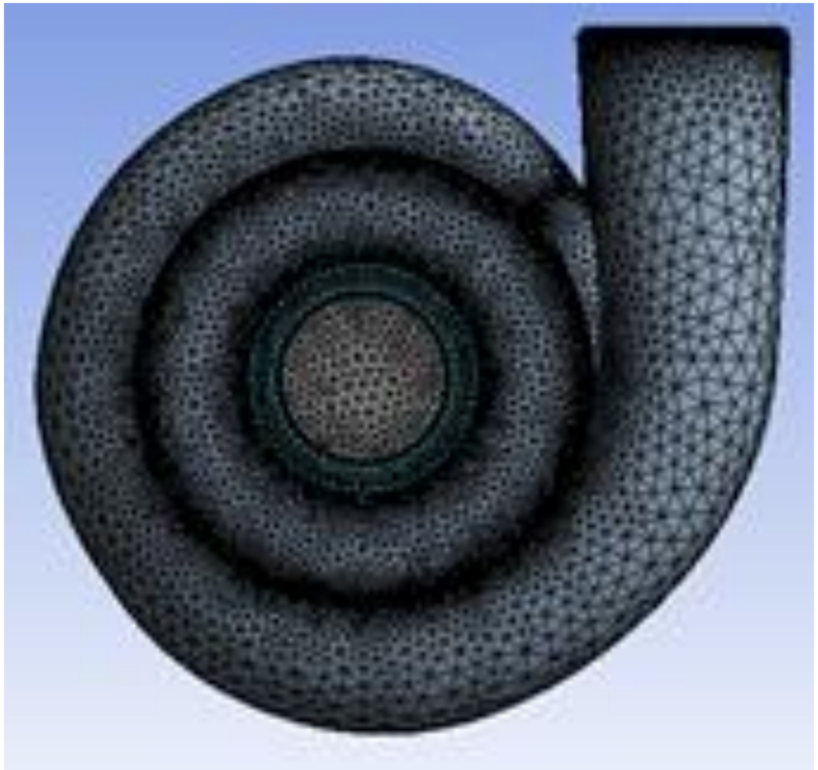
- Meshing for FRANCIS TURBINE can be done by direct meshing and meshing by areas. In this study, meshing is divided into three areas of fluid model; casing, runner and draft tube. Meshing of these areas is shown in figure.

GENERATION OF MESHING :

WITHOUT DRAFT TUBE :

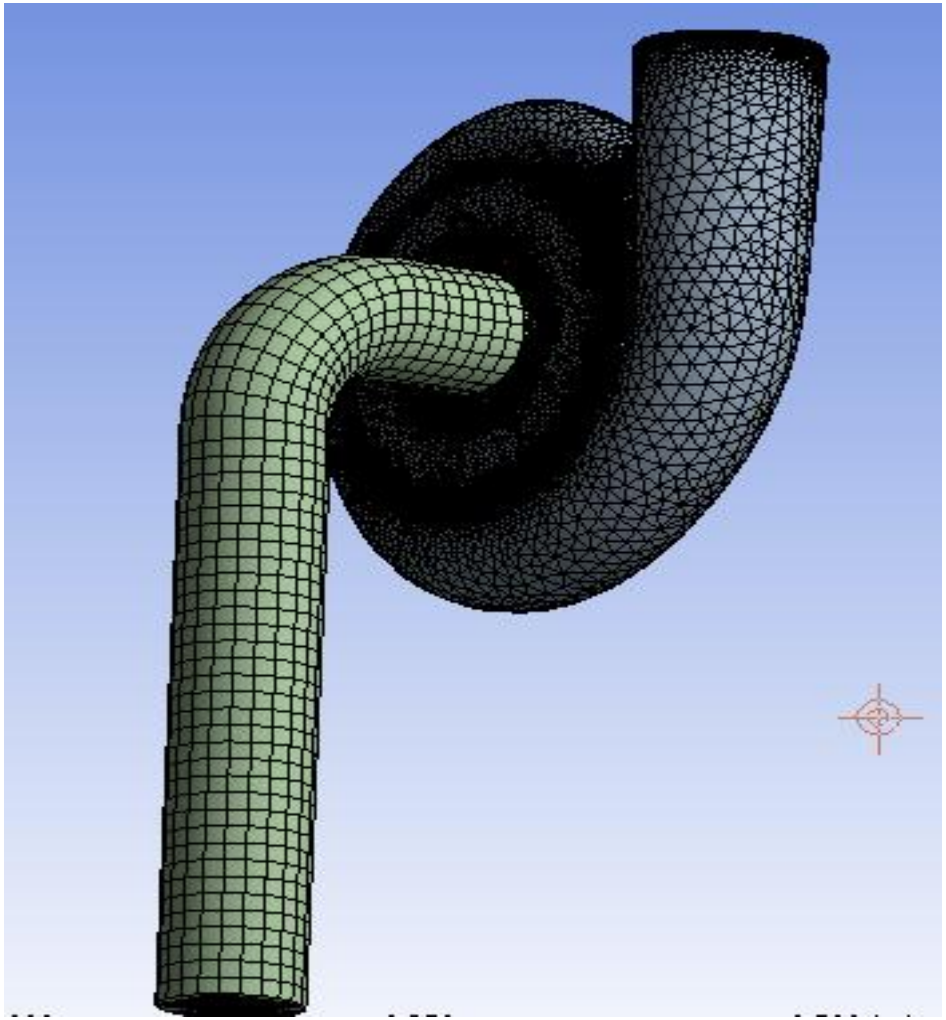


WITHOUT GUIDE VANE



WITH GUIDE VANE

GENERATION OF MESHING :



WITH GUIDE VANE AND DRAFT TUBE :

SOLUTION PROCEDURE IN CFD :

Following are the steps performed in CFX while solving :

- **Step 1 : Flow analysis** :-
- Analysis type: Fluid model steady state analysis type is considered.
- Define domain: Present fluid model has three fluid domains i.e. impeller, casing and draft tube. For casing and draft tube, stationary domain motion is considered. For impeller, rotating type domain motion is considered. No heat transfer, No slip condition and smooth wall function is considered for three fluid domains.
- Boundary conditions: Boundary conditions are defined on inlet of casing and outlet of draft tube.

SOLUTION PROCEDURE IN CFD :

- **Step 2 : Interface conditions :-**

- Interface conditions :

  First interface is defined in between casing and impeller. In this outlet of casing and inlet of impeller are interfaced to each other.

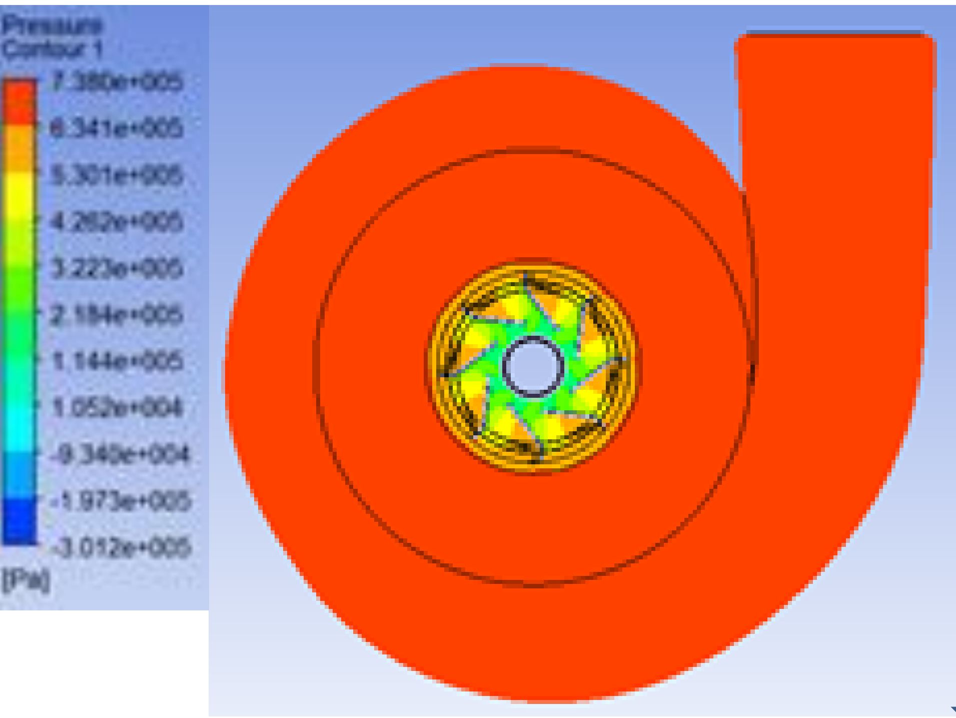
  Second interface is defined in between impeller and draft tube. Here interfacing is done between outlet of impeller and inlet of draft tube.

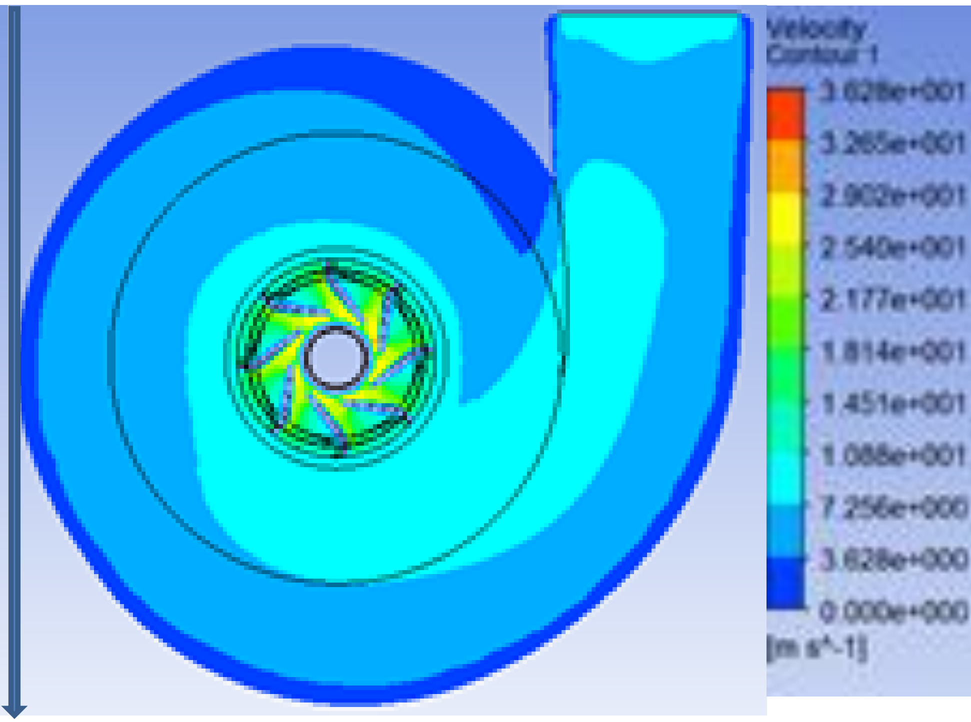
Step 3 : Solver :-

- In basic solver controls, high resolution for advection term and first order turbulence numeric is considered.

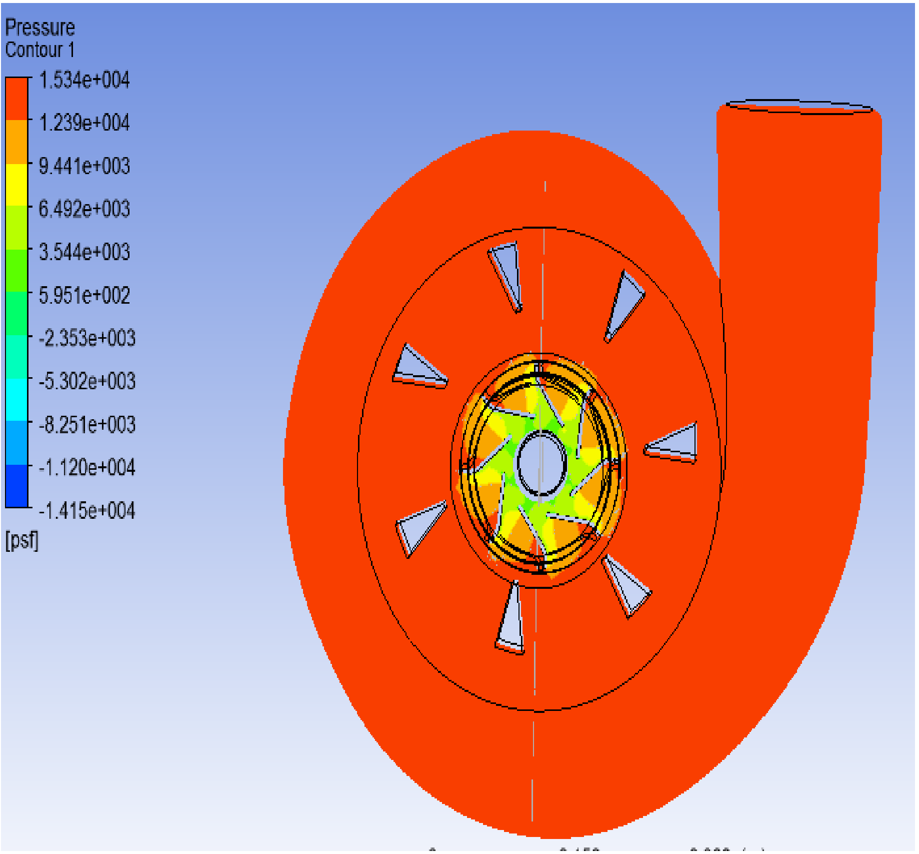
SOLUTION IN CFD :

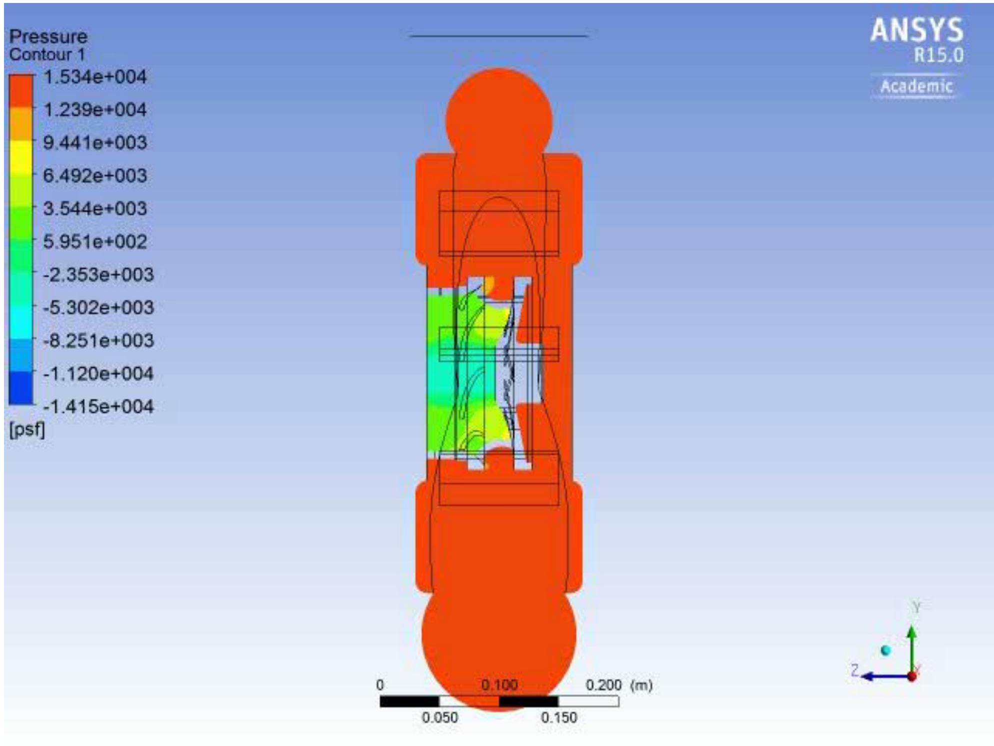
WITHOUT GUIDE VANE :



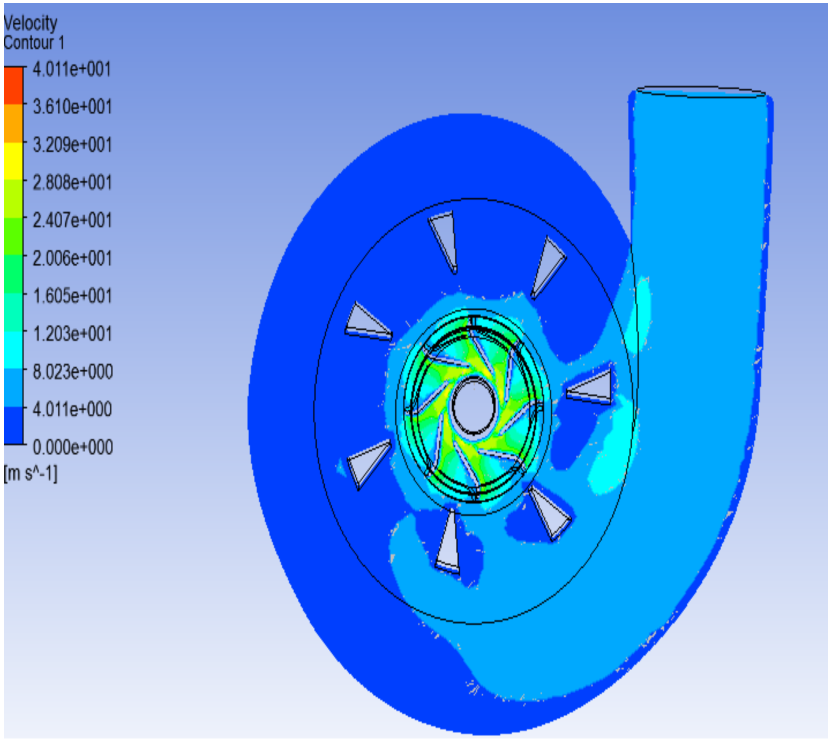


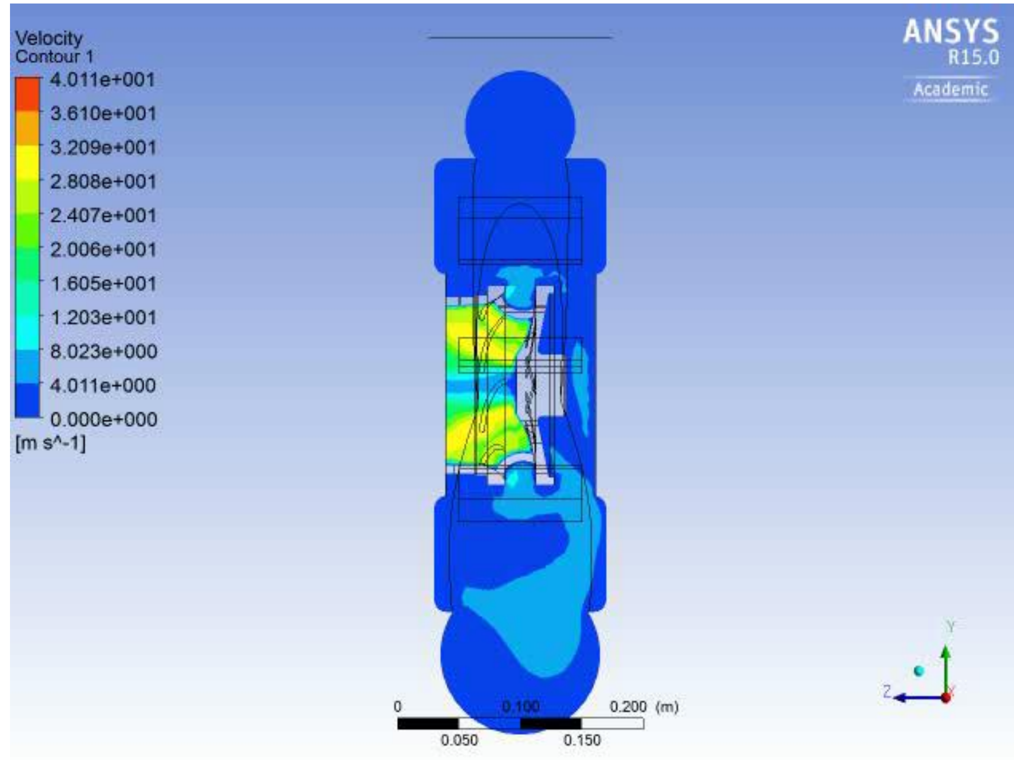
WITH GUIDE VANE :



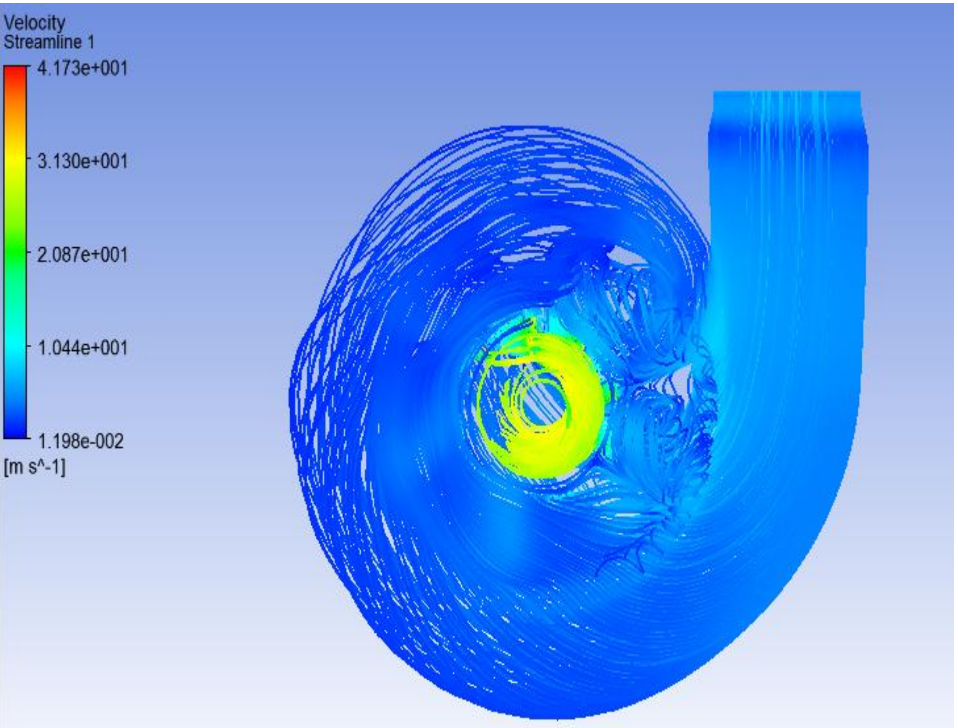


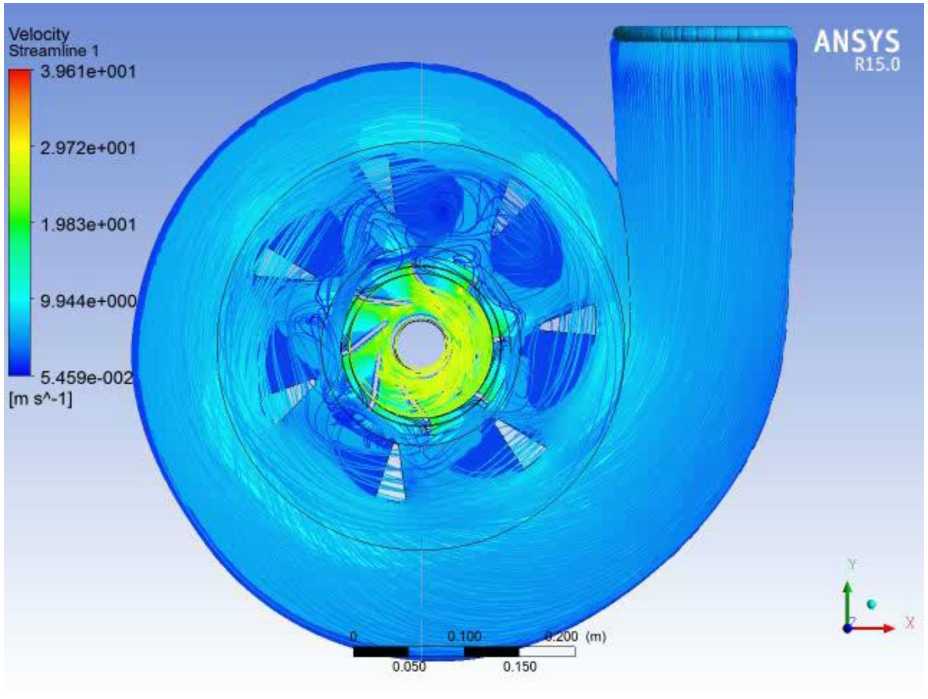
WITH GUIDE VANE :



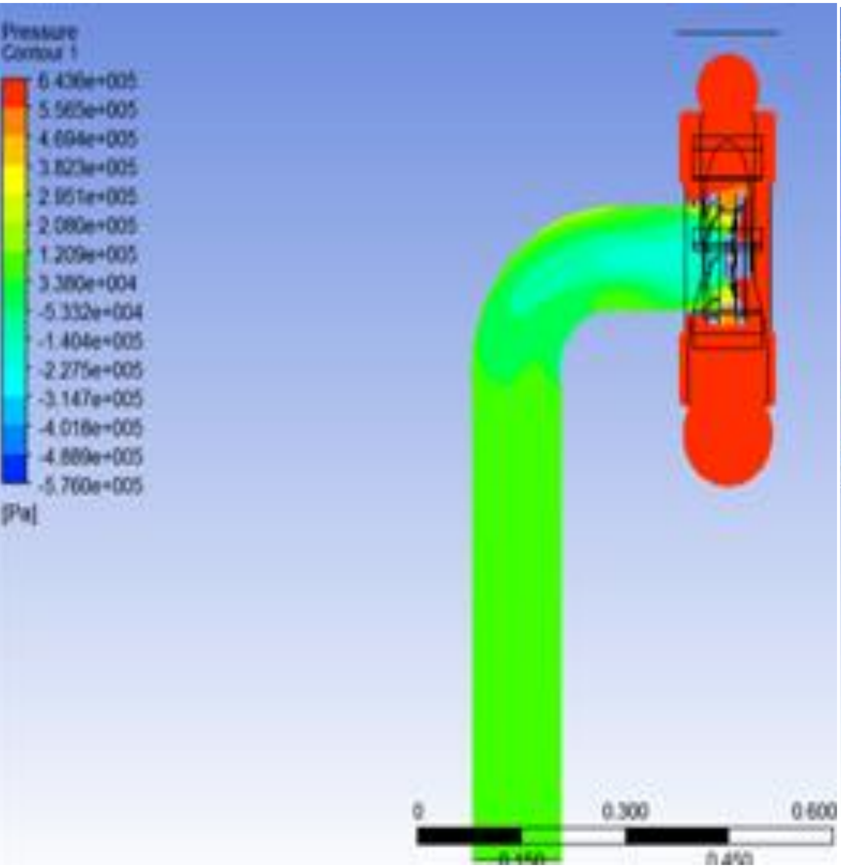


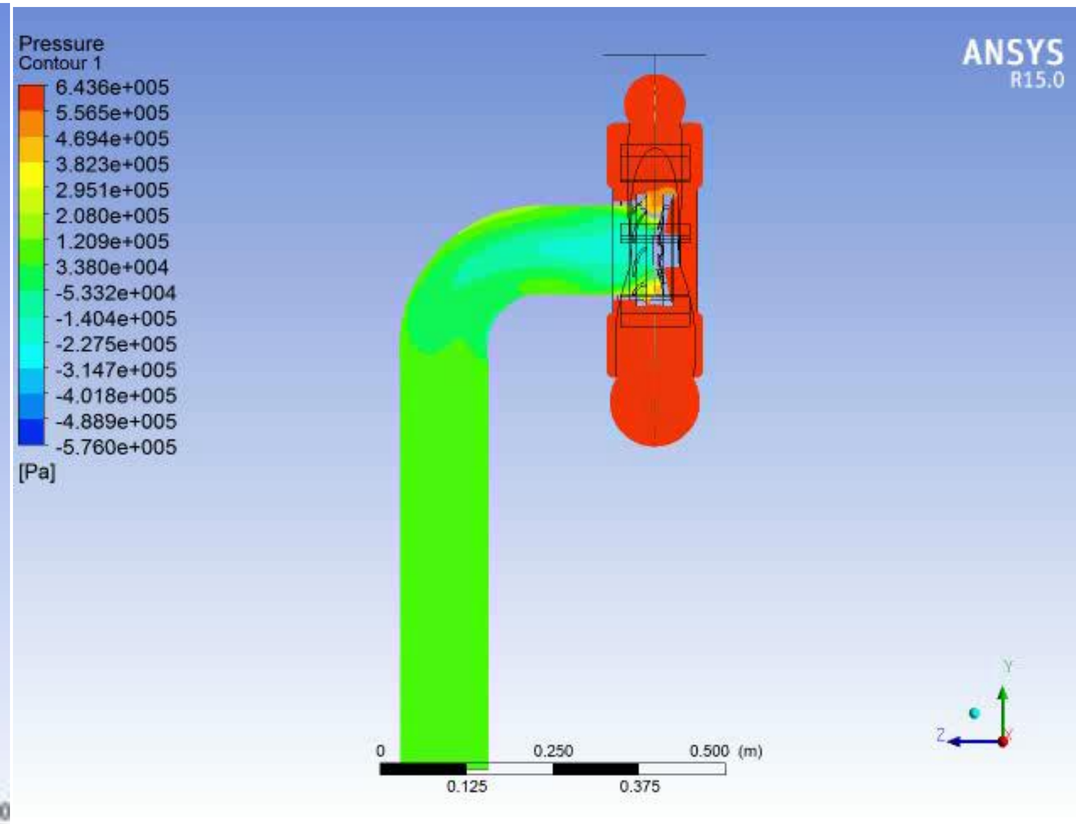
WITH GUIDE VANE(along with stream line) :



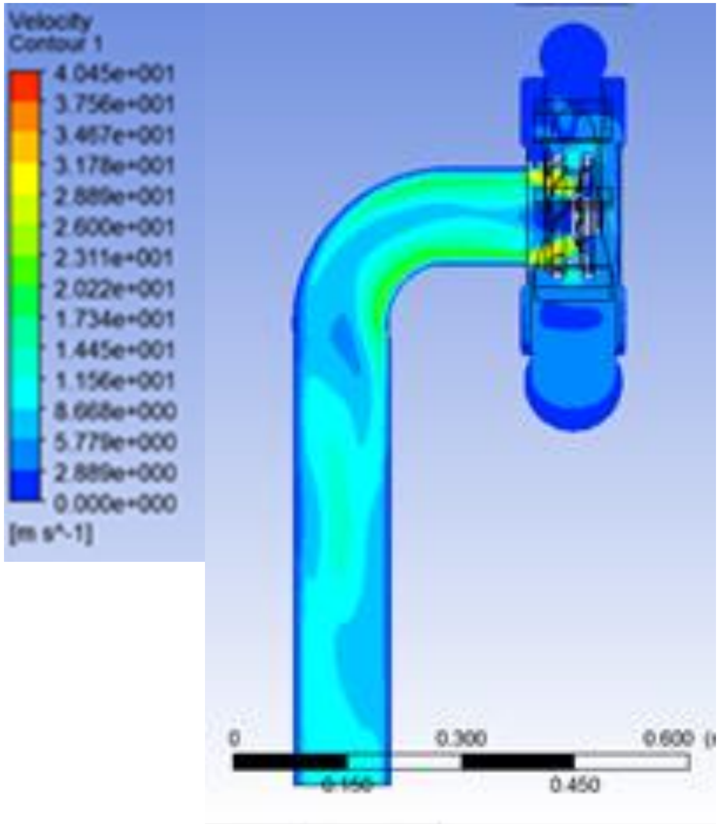


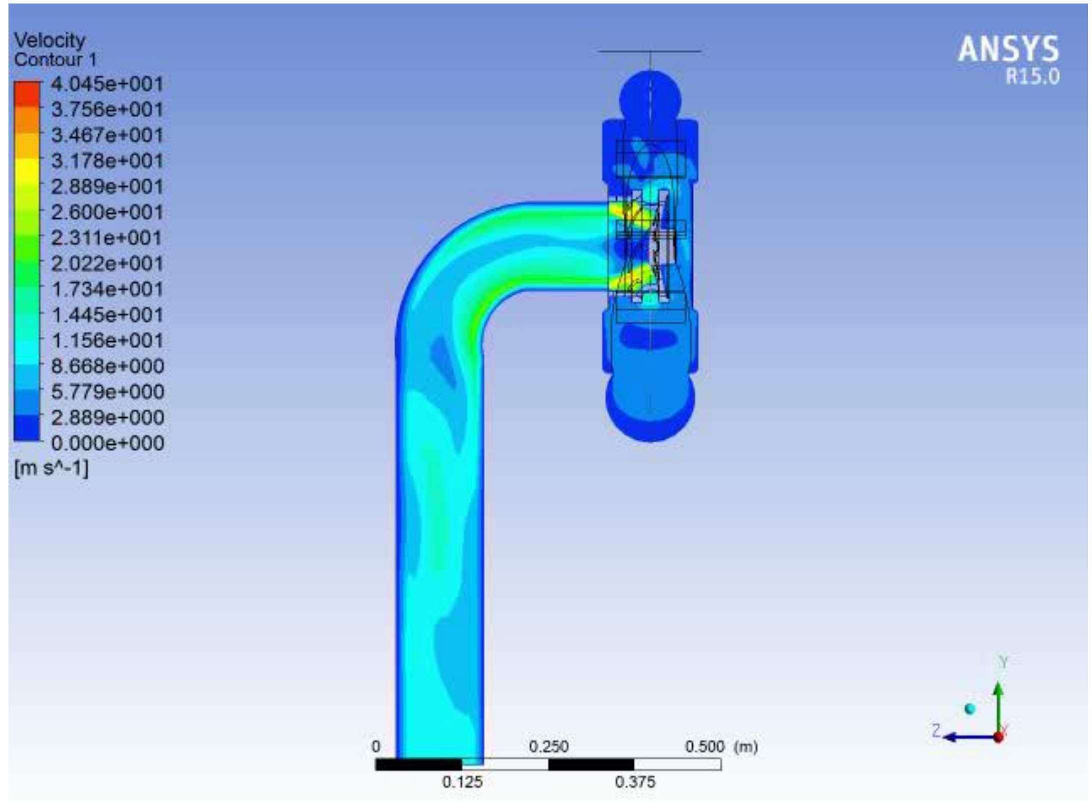
WITH GUIDE VANE AND DRAFT TUBE :





WITH GUIDE VANE AND DRAFT TUBE :





CONCLUSION :

- CFD can be a valuable and accurate tool for evaluating the head loss, cavitations zones and the resulting reduction in plant output associated with poor performing hydraulic structures.

- In this study , for that preparation of fluid model of casing and runner is made accurate at all locations where analysis done.

- Simulation facilitated to reduce cost and time of the experiments. If once model of francis turbine is simulated then it would be easy to check the performance of the francis turbine at any condition.

- After generating model of turbine ,we can put any type of pressure contour and velocity contour as well as temperature ,mass-flow rate as per requirement of application and find expected level of calculations.

RESEARCH PAPERS :

1. Sanjay Jain, R. P. Saini and Arun Kumar, "**CFD approach for prediction of efficiency of francis turbine** " Oct 21-23, 2010.

2. Krishna Prasad Shrestha, Bhola Thapa, Ole Gunnar Dahlhaug, Hari P. Neopane and Biraj Singh Thapa, " **Innovative design of francis turbine for sediment laden water** " TIM International conference - 2012.

3. R.A. Saeed , A.N. Galybin and V. Popov, "**Modelling of flow-induced stresses in a Francis turbine runner** " advances in engineering software 41 (2010) 1245–1255.

4. R.A. Saeed and A.N. Galybin, "**Simplified model of the turbine runner blade** " Engineering Failure Analysis 16 (2009) 2473–2484.

5. XIAO Ruofu , WANG Zhengwei and LUO Yongyao, “**Dynamic stresses in a francis turbine runner based on fluid-structure interaction analysis** ” October 2008.

6. Z. Čarija and Z. Mrša “ **Complete francis turbine flow simulation for the whole range of discharges”,** September, 2003.

7. Ravindra R. Navthar1 Joshi Tejas2, Dhaneshwar Saurabh2 ,Domale Nitish2, Abhyankar Anand2 ”**CFD analysis of francis turbine”,** July 2012.

8. Radu Negru1,*, L. Marsavina1 and Seby Muntean2, “**Analysis of flow induced stress field in a francis turbine runner blade”,** May , 2011.

9. R.A. Saeed , A.N.Galybin , V.Popov, “**3D fluid–structure modelling and vibration analysis for fault diagnosis of Francis turbine using multiple ANN and multiple ANFIS**” Mechanical Systems and Signal Processing 34 (2013) .

10. Hyen-Jun Choi , Mohammed Asid Zullah , Hyoung-Woon Roh , Pil-Su Ha , Sueg-Young Oh, Young-Ho Lee , “**CFD validation of performance improvement of a 500 kW Francis turbine**” Renewable Energy 54 (2013) .

11. Yongzhong Zeng, Xiaobing Liua, Huiyan Wang, **“Prediction and experimental verification of vortex flow in draft tube of Francis turbine based on CFD”** Procedia Engineering 31 (2012) .

FUTURE WORK-PLAN (2013-14) :

	7^{th} sem	JAN	FEB	MAR
Literature review				
Dismantle the model				
Take dimensions of all parts of model				
Generating the drawing of model in PRO-E				
Generating the fluid model of casing & runner				
Based on research paper apply the boundary condition in CFD analysis of model				
Calculate pressure counter, velocity counter &flow analysis of model				
Result analysis & discussion				
Future scope & conclusion				

Thank you . . .