Benchmarking of Open FOAM for CFD applications

Kalash Sarode[1], Radhika Naik[2], Shantanu Jagtap[3], Aditya Pachpor[4], V.H. Bansode[5]

[1]UG student, Mechanical dept., Smt. Kashibai Navale College of Engg, **kalshsarode@gmail.com**
[2]UG student, Mechanical dept., Smt. Kashibai Navale College of Engg, **radhikan785@gmail.com**
[3]UG student, Mechanical dept., Smt. Kashibai Navale College of Engg, **shantanujagtap173@gmail.com**
[4]UG, Mechanical dept., Smt. Kashibai Navale College of Engg, **paditya63@gmail.com**
[5]Assistant Professor, Mechanical dept., Smt. Kashibai Navale College of Engg.

ABSTRACT

As a substitute of costly experimentation methods, CFD has become popular in the industrial scenario. Most of the commercial software are expensive even though they have added advantage of user-friendly GUI, ease of learning and use for beginners. The main objective is to study OpenFOAM which is a C++ tool released as free and open source software. The purpose of this project is to identify/observe and determine the performance of OpenFOAM for CFD simulation program after the 3D design and modelling of various systems provided. The main system chosen for validation of the results is a centrifugal pump used for common industrial applications. Basically, this project revolves around the idea of investigating the effect and distribution of velocity profile and pressure within the pump. A method has been created using the OpenFOAM solver simpleFoam using both stationary and rotating mesh domains to allow relative motion between inflow/outflow and impeller blades. The cyclic AMI was used as a boundary condition for the patches to allow simulation between them. Meshing was done with the commercial tool by ICEM CFD. 3D Navier-Strokes equation were solved using ANSYS Fluent. The standard k-ω SST model chosen for turbulence model. Along with the pump, a few type systems and other basic systems were considered to study and master OpenFOAM. The results obtained are compared to check the reliability of OpenFOAM. The required output of the project is to benchmark the open source software against commercial tools for applications of Computational Fluid Dynamics.

Keywords*: Pump, CFD analysis, simulation, ANSYS Fluent, OpenFOAM, pressure and velocity distribution, simpleFoam.*

---

1. INTRODUCTION

Computational fluid dynamics (CFD) is a branch of fluid mechanics that uses numerical analysis and data structures to solve and analyze problems that involve fluid flows. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. Ongoing research yields software that improves the accuracy and speed of complex simulation scenarios such as transonic or turbulent flows. Initial experimental validation of such software is performed using a wind tunnel with the final validation coming in full-scale testing, e.g. flight tests.

CFD analysis reduces development time, minimizes costs and increases the reliability of prototype designs. CFD simulations can be used to model fluid flows over a wide range of physical scales. The benefit of using CFD is that we get better details, better prediction, can design better and faster, economical, meet environmental regulations and ensure industry compliance. CFD has a wide range of applications in Aerospace, Automotive, Biomedical, Chemical processing, HVAC, Hydraulics etc. the objective of this paper is to study.

OpenFOAM (for "Open source Field Operation and Manipulation") is an open-source software meaning totally free, no licensing fees, unlimited number of users, jobs & cores.

The company OpenCFD, the main developer of OpenFOAM, was bought by ESI, which is committed to continue funding the development, while maintaining the open source nature of this technology. This eliminates the fear of problem support. In the right hands OpenFOAM gives very accurate results.It has ability to create individualized solutions. Unlike commercial software, OpenFOAM offers great scope for custom development. It has ability to quickly achieve accurate results. Refining of results compared with experiment is significantly faster.With OpenFOAM, statistics prove 40% reduction in costs associated with CFD.This software has the power can revolutionize the CFD industry. It can replace all the commercial software that are used.It reduces the cost of product indirectly. It provides many different environment and many different environments can be made to suit our conditions. Hence it can be used anywhere and freely.

In all of the CFD approaches the same basic procedure is followed.

- During preprocessing the geometry and physical bounds of the problem can be defined using computer aided design (CAD). From there, data can be suitably processed (cleaned-up) and the fluid volume (or fluid domain) is extracted
- The volume occupied by the fluid is divided into discrete cells (the mesh). The mesh may be uniform or non-uniform, structured or unstructured, consisting of a combination of hexahedral, tetrahedral, prismatic, pyramidal or polyhedral elements.
- The physical modeling is defined – for example, the equations of fluid motion + enthalpy + radiation + species conservation
- Boundary conditions are defined. This involves specifying the fluid behavior and properties at all bounding surfaces of the fluid domain. For transient problems, the initial conditions are also defined.
- The simulation is started and the equations are solved iteratively as a steady-state or transient.
- Finally a postprocessor is used for the analysis and visualization of the resulting solution.

2. GOVERNING PHYSICS

2.1 Discretization Methods

The stability of the selected discretization is generally established numerically rather than analytically as with simple linear problems. Special care must also be taken to ensure that the discretization handles discontinuous solutions gracefully. The Euler equations and Navier–Stokes equations both admit shocks, and contact surfaces.

Some of the discretization methods being used are:

1. Finite Volume Method
2. Finite element method
3. Finite difference method

2.2 Turbulence Model used: K-ω Model

It is a two transport-equation model solving for kinetic energy k and turbulent frequency ω. This is the default k–ω model. This model allows for a more accurate near wall treatment with an automatic switch from a wall function to a low-Reynolds number formulation based on grid spacing. Demonstrates superior performance for wall-bounded and low Reynolds number flows. Solves one equation for turbulent kinetic energy k and a second equation for the specific turbulent dissipation rate (or turbulent frequency) ω. This model performs significantly better under adverse pressure gradient conditions. The model does not employ damping functions and has straightforward Dirichlet boundary conditions, which leads to significant advantages in numerical stability. It has superior performance for wall-bounded boundary layer, free shear, and low Reynolds number flows. Suitable for complex boundary layer flows under adverse pressure gradient and separation. It can be used for transitional flows. It requires mesh resolution near the wall.

The eddy viscosity ν_T, as needed in the RANS equations, is given by: $\nu_T = k/\omega$, while the evolution of k and ω is modeled as:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho u_j k)}{\partial x_j} = \rho P - \beta^* \rho \omega k + \frac{\partial}{\partial x_j}\left[\left(\mu + \sigma_k \frac{\rho k}{\omega}\right)\frac{\partial k}{\partial x_j}\right], \qquad \text{with } P = \tau_{ij}\frac{\partial u_i}{\partial x_j},$$

$$\frac{\partial(\rho \omega)}{\partial t} + \frac{\partial(\rho u_j \omega)}{\partial x_j} = \frac{\gamma \omega}{k} P - \beta \rho \omega^2 + \frac{\partial}{\partial x_j}\left[\left(\mu + \sigma_\omega \frac{\rho k}{\omega}\right)\frac{\partial \omega}{\partial x_j}\right] + \frac{\rho \sigma_d}{\omega}\frac{\partial k}{\partial x_j}\frac{\partial \omega}{\partial x_j}.$$

3. CFD ANALYSIS

The process includes geometry and meshing, selection of physics and fluid properties, specification of boundary conditions, initialization and solution control, monitoring convergence, result plotting. ICEM is used for geometry and meshing and same mesh file is used for both OpenFOAM and ANSYS FLUENT, as ICEM gives better control over mesh generation and quality.

ISSN: 2321-8169
1468 – 1474

Given:

- Geometry (of pump)
- Inlet volume flow rate - 300 lpm
- Outlet Pressure - atmospheric pressure
- rpm of rotating body (impeller) - 2900 rpm

Assumptions:

- Steady-state condition
- Constant fluid properties
- Incompressible fluid flow
- Walls are considered to be smooth, hence any disturbance due to roughness factor of the walls is to be neglected.

3.1 Pre-processing

The original geometry is imported a .stp file in ICEM. The centrifugal pump is axial in radial out with water as working fluid.

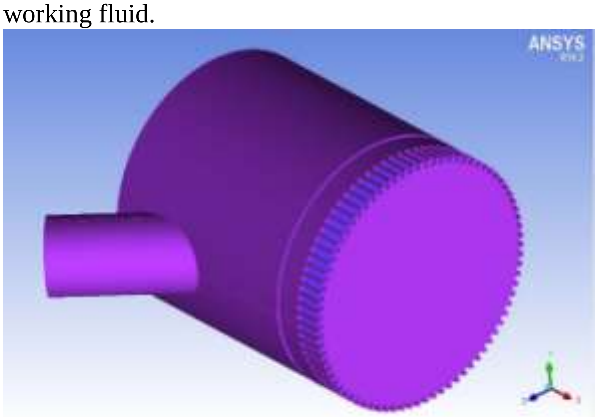

Fig-1: Original geometry of pump

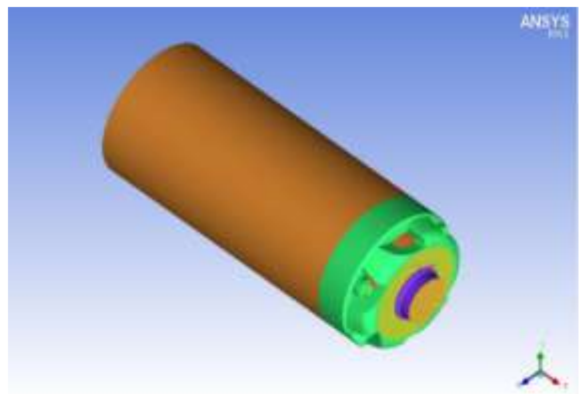

Fig-2: Pump Geometry with the extended outlet.

ICEM CFD gives the best control over mesh refinement and the mesh quality. Tetrahedral mesh is used. All the surfaces are assigned a suitable mesh size and mesh is computed. by assigning finer size in the identified regions of bad elements, mesh is again computed by trial and error method until there is absence of bad elements.

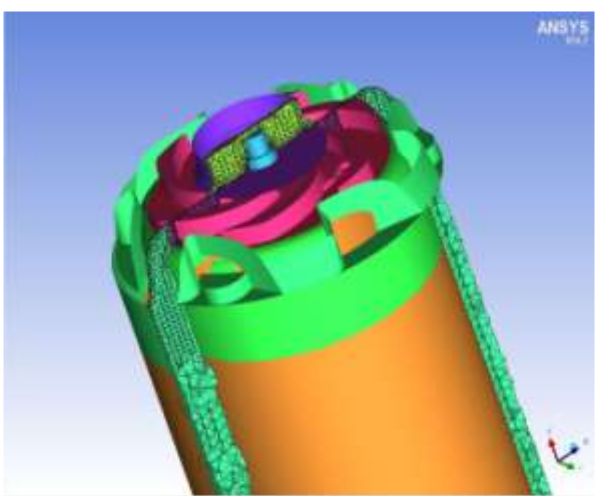

Fig-3: Pump Mesh cut-plane view (Z-axis)

--- Mesh Info ---
Element types :
NODE : 292
LINE_2 : 11983
TETRA_4 : 1555413
TRI_3 : 259159
Element parts :
EXTENDED : 25197
IMPELLER : 109530
IMPELLER_INLET : 3008
INLET : 1101
INLET_VOL : 5535
INNER_CYLINDER : 14260
MRF-BOTTOM : 42453
MRF-CYLINDER : 11025
MRF-STATIONARY-WALL : 15829
MRF_IN : 4939
OUTER_CYLINDER : 38056
OUTLET2 : 501
VOL_INLET : 94600
VOL_MRF : 926587
VOL_OUTLET : 534226
Total elements : 1826847
Total nodes : 323562
Min : -426.5 -87.2489 -87.25
Max : 16 87.2498 87.25

Fig-4: Pump Mesh info.

Boundary Conditions Foam uses "cyclicAMI" for rotation of volume which needs over lapping surfaces surrounding the rotating volume. AMI are the surfaces created by the user for the specific use. It may lie in fluid domain itself. Hence it is necessary to define these surfaces as interiors which will then let the fluid pass through it.
Inlet boundary condition: *Constant mass-flow inlet* was used.
Outlet boundary condition: *Constant pressure-outlet* was used.
Walls: *No-slip*

The values for initialization for $k - \omega\ SST$ model are calculated using the following formulae

$Re = \rho * u * Dh/\mu$; $uavg = \mu * Re/(\rho * Dh)$;

$k = (3/2) * (uavg * I)^2$; $I = 0.16 * (Re)^{-1/8}$;

$l = 0.007 * Dh$ $\omega = {C_\mu}^{-1/4} * k^{1/2}/l$;

(C_μ= 0.09 in all cases)

Where,

Re = Reynolds number ; C_μ = turbulence model constant

ρ = Density of fluid ; l = turbulent mixing length

$uavg$= average velocity at the inlet patch ; k = turbulent energy

Dh = hydraulic (inlet) diameter ; I = turbulence intensity

ω = specific turbulent dissipation rate ; μ = Viscosity

3.2 Solving Process

OpenFOAM
The k-ω is used, so the convergence criteria (tolerance values and other parameters alike were set in the 0/fvsolution (iterative solvers for separate variables) and 0/fvschemes (order of accuracy and method).
The convergence criteria will be satisfied if any one of the following conditions are reached (and the solver stops):

- the residual falls below the solver tolerance, tolerance;
- the ratio of final residual to initial residuals fall below the, relTol;
- the number of iterations exceeds a maximum number of iterations, maxIter;

simpleFoam solver was selected from the list of incompressible flow solvers. Before executing the solver, following command was used to document the residuals numerically in a "log" file :
simpleFoam > log &
Then the simpleFoam command was used to run the simulation. After running the results were viewed in paraview by using the paraFoam command.

ANSYS FLUENT
After meshing, the type of the solver needed (steady, unsteady, density based, pressure based.) to address the specific problem was decided. Later an appropriate physical model was selected (turbulence, multiphase, combustion). This was followed by defining the material properties operation conditions and boundary conditions. Convergence models or residual models were established to reduce unnecessary computing time. The flow field is initiated after all this setup with approx. initial assumptions.

3.3 Post Processing

Post processing in OpenFOAM uses different software. Paraview is used for visual representation of results obtained from OpenFOAM. The plots in OpenFOAM are done with the use of log file created (Numerical documentation of all the simulation data.

- Residuals in OpenFOAM

Using gnuplot we can plot residuals vs. no. of iterations on log scale. The difference between value of parameter in successive iterations is called residuals. Thus, gnuplots for P, U, ω & k are obtained.

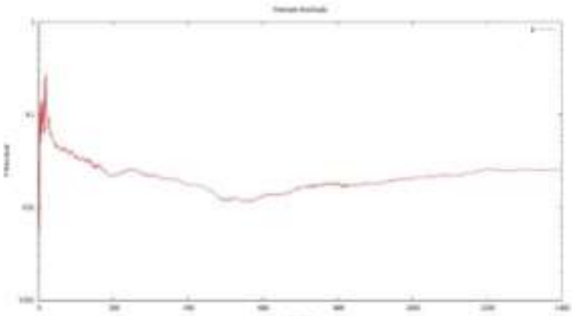

Fig-5: p-Residual

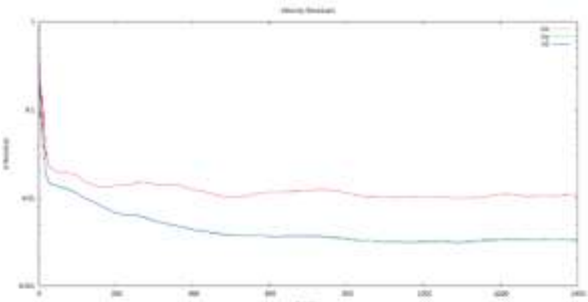

Fig-6: u- Residual

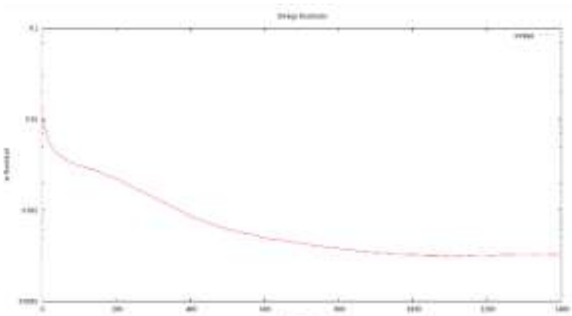

Fig-7: k-Residual

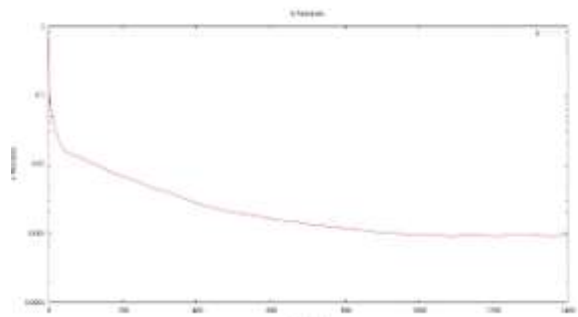

Fig-8: ω -Residual

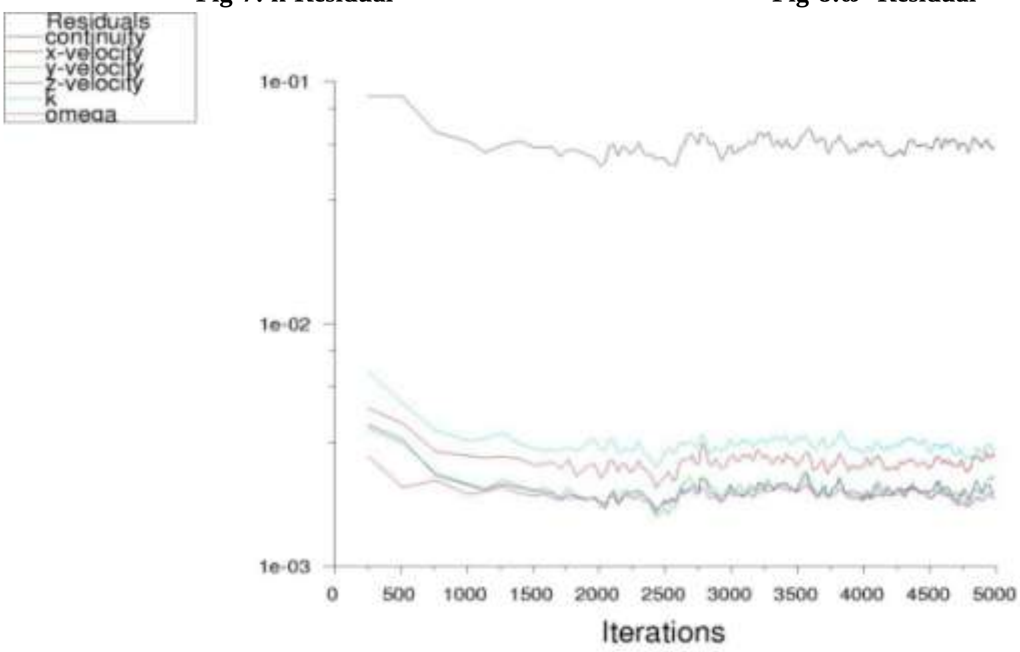

Fig-9:Residuals in FLUENT

It is observed that all the residuals are decreasing (or tending to), and are becoming constant with the number of iterations showing that the result is converging.

ISSN: 2321-8169
1468 – 1474

3.3.1 Pressure Contour Plots

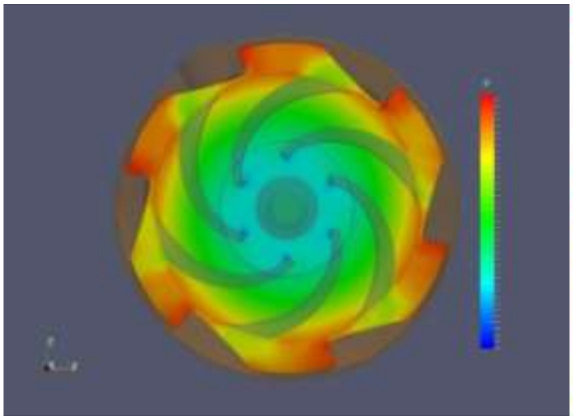

Fig-10: Pressure plot 1 in OpenFOAM

Fig-11: Pressure plot 1 in FLUENT

The pressure contours are of OpenFOAM and of Fluent match at first glance (increasing radially) but then in a closer look, the plane cut in OpenFOAM shows a more smoothening contour radially but in Fluent, the pressure is tending to increase in the direction of impeller blade.
The max pressure contours are concentrated at the outflow of fluid from VOL_MRF.
The minimum pressure contours are mostly observed towards the inner blade.

3.3.2 Velocity Contour Plots

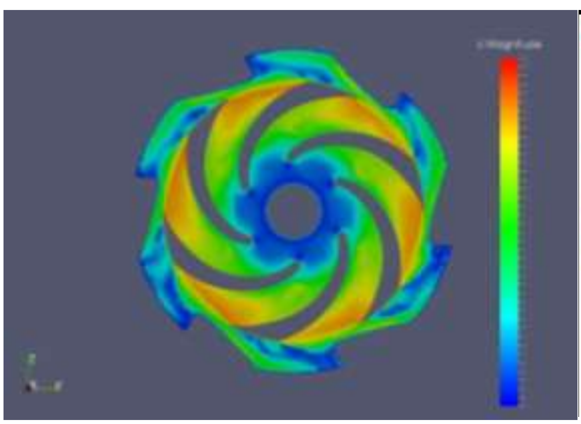

Fig-12: Velocity plot in OpenFOAM

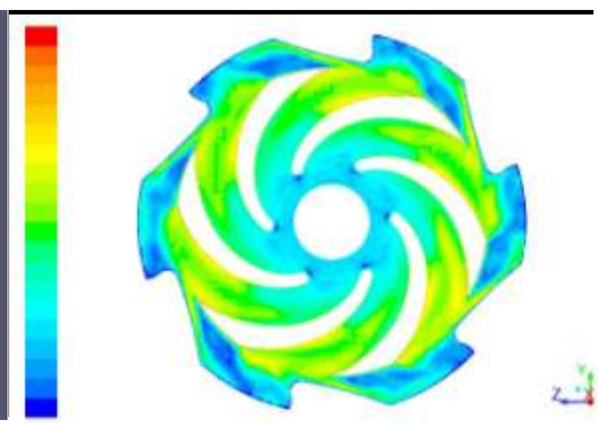

Fig-13: Velocity plot 1 in FLUENT

The velocity contours of OpenFOAM and Fluent match to a great extent in both considered cut planes. For impeller cut, velocity is radially increasing (basic logic also points towards it.) It can be safely stated that velocity contours of OpenFOAM are more similar in nature when compared with Fluent than pressure contours as seen above.

3.3.3 Pressure Differences (Area Weighted Averages)

The differences between area weighted averages of pressure is a good criteria for comparison between OpenFOAM and ANSYS Fluent. The negative sign indicates that the pressure is increasing towards the fluid out patch everywhere. Hence the nature of fluid flow is as follows in both softwares :

Inlet > MRF_IN > MRF_CYLINDER > Outlet.

Main info		Differences		
Related Vol	**Surfaces**	**OpenFOAM**	**Fluent**	**Relative**
VOL_INLET	(Inlet) - (MRF_IN)	-4979.6 Pa	-90.57 Pa	4889.03
VOL_MRF	(MRF_IN)- (MRF_Cylinder)	-124493.15 Pa	-107966.83 Pa	-16526.32
VOL_OUTLET	(MRF_Cylinder) - (Outlet)	-15873.25 Pa	-16561.13 Pa	687.88

4. CONCLUSION

From both the simulations of OpenFOAM and ANSYS FLUENT it is concluded that there is a slight reverse flow through some faces of the outlet. This reverse flow influences the properties of the domain .The results of OpenFOAM and ANSYS Fluent are similar. They show same behavior of pressure, velocity, k and omega but there exists a slight difference in magnitudes of these properties

Due to different discretization techniques there might be a variation in the results of the solver. There would be improvement in OpenFOAM results if more simulation trials are conducted. Also the simulation time would be greatly reduced if parallel processing is used. Parallel processing is quite complex and requires advanced knowledge of processing in OpenFOAM. Thus OpenFOAM is tedious and time consuming compared to ANSYS Fluent. OpenFOAM allows us to create our own environment and has better boundary conditions and options hence it has a greater scope in future.

OpenFOAM, when in experienced hands, is good for CFD simulations and can be used for commercial purposes. Since it is a free, open source software there in more scope for development of complex geometries, domains and fluid flow problems that cannot be studied experimentally.

5. REFERENCES

[1] Experimental and CFD Analysis Of Centrifugal Pump Impeller- A Case Study Mechanical Engineering, SPCE, Visnagar, India.

[2] Design and CFD Analysis of Centrifugal Pump, Volume 3, Issue 3, May-June, 2015 ISSN 2091-2730 668, IJERGS.

[3] CFD Analysis of Domestic Centrifugal Pump for Performance Enhancement, (IRJET) e-ISSN: 2395-0056 Volume: 02 Issue: 02 | May-2015 www.irjet.net p-ISSN: 2395-0072.

[4] Dynamic flow analysis using an OpenFOAM based CFD tool: Validation of Turbulence Intensity in a testing site. Livio Casella.

[5] CFD Analysis of centrifugal pump impeller for performance enhancement P.Gurupranesh, R.C.Radha, N.Karthikeyan.IOSR-JMCE)

[6] https://openfoamwiki.net/index.php/Main_Page