RESEARCH ARTICLE OPEN ACCESS

Flow analysis of centrifugal pump using CFX solver and remedies for cavitation mitigation

Dr. G. Rambabu*, S. Sampath** G. Karthik*** S. Siva Teja****
*(Department of Mechanical Engineering, Andhra University College of Engineering, A.U.C.E(A), Visakhapatnam-530003
** (Department of Mechanical Engineering, Andhra University College of Engineering, A.U.C.E(A), Visakhapatnam-530003
***Department of Mechanical Engineering, Andhra University College of Engineering, A.U.C.E(A), Visakhapatnam-530003
*****Department of Mechanical Engineering, Andhra University College of Engineering, A.U.C.E(A), Visakhapatnam-530003

ABSTRACT
In this scholarly thesis pertinent to the working of centrifugal pump, a CFD solver namely CFX is employed in order to simulate fluid flow characteristics with well-defined constraints and boundary conditions defining the problem. Stringent solid model is meticulously prepared encompassing the present day usage and constructional features of a centrifugal pump and is constrained with various boundary conditions having fixed domain in order to evaluate plots and results. To spearhead and facilitate this analysis program a numerical approximation tool with high degree of convergence rate called ANSYS 15.0 software is used. The ASNYS software avoids tedious calculations presumably impending in the design procedure and uses ultimate numerical tool to approximate the solution of the partial differential equations associated with continuity, momentum and energy phases of a flow problem in a 3-D model. This exquisite feature of ANSYS enables designer to optimize the design procedure in an iterative manner based on the final plots of post-processing phase. In addition, the scholarly writing also constitutes the appraisal of the most debilitating and painstaking problem retarding the efficiency of the centrifugal pump known as cavitation. Possible remedies for overcoming this problem will be indirectly inferred from the various plots and figures derived from the post-processing phase of the design process.
***Keywords* -** Cavitation, NPSH, CFD.

I. Introduction
(What is cavitation?)

Cavitation is the formation and subsequent collapse or implosion of vapor bubbles inside the pump. It occurs because the absolute pressure of the liquid drops below the liquid's vapor pressure. When vapor bubbles collapse with enough frequency, it sounds like marbles and rocks are moving through the pump. If larger number of vapor particles accumulate they become irresistible and causes indentation on the surface encompassing the pump leading to pitting action. This pitting action corrodes the metal leading to noisy operation of the pump and sometimes leads to massive explosion. Cavitation is a painstaking problem in today's commercial usage of pump and must be avoided to avert any retardation in the functioning of the pump. Cavitation is bound to lower the efficiency of the pump. If the pump operates under cavitating conditions for rough times, the following can occur:

- Pitting marks on the impeller blades and on the internal volute casing of the pump.
- Premature bearing failure.
- Shaft breakage and other fatigue failures in the pump.
- Pre-mature mechanical failure.

Factors perpetrating cavitation in a pump:

- Fall of absolute pressure at the suction nozzle below the saturation vapor pressure corresponding to ambient temperature of the water.
- An increase in the temperature of the pumped liquid.
- An increase in the velocity or flow of the fluid.
- Separation and reduction of the flow due to a change in the viscosity of the liquid.
- Undesirable flow conditions caused by obstructions or sharp elbows in the suction piping.
- The pump is inadequate for the system.

II. Estimation and prediction of cavitation

A powerful tool to access the demerits of a centrifugal pump is the net positive suction head denoted as NPSH. This tool helps in the evaluation of the toughest pain staking problem entangled in the operating condition of the centrifugal pump. Net positive suction head is defined as the minimum head required at any point of the centrifugal pump to avoid cavitation. It is the energy in the liquid required to overcome the friction losses from the suction nozzle to the eye of the impeller without causing vaporisation. It is a characteristic of the pump and is indicated on the pump's curve. It varies by design, size, and the operating conditions. It is determined by a lift test, producing a negative pressure in inches of mercury and converted into feet of required NPSH. According, to the standards of hydraulic institute, a suction lift test is performed on the pump and the pressure in the suction vessel is lowered to the point where the pump suffers a 3% loss in the total head. This point is called the NPSHr of the pump.

Generally, this net positive suction head theory is only applicable to the eye and suction of the impeller only because the vapor pressure at running conditions is highly capricious due to the advent of the frictional head loss and its conversion into heat energy leading to a rise in the temperature of the liquid. This ultimately decreases the vapor pressure. Moreover, the vapor pressure distribution is not stagnant throughout the impeller plane and thus NPSH is not applicable to the total portion of the pump.

$$NPSH = \frac{p1-pv1}{pg}+\frac{C^2}{pg} \qquad (1)$$

where,
p1- absolute pressure at inlet.
pv1- vapor absolute pressure.
C- velocity at inlet.
p- density of liquid.
g- acceleration due to gravity.
and moreover,

$$NPSHr = NPSHa + 3\ (feet) \qquad (2)$$

where, NPSHr- net positive suction head required.
NPSHa- net positive suction head available.
Following are the exquisite features of NPSH:

- Prediction of cavitation and its evaluation.
- Determining the necessary NPSH inorder to overcome cavitation at all working conditions of the pump.
- Evaluating the required NPSH by adding 3 feet head to the available NPSH.

III. What is CFD?

Computational fluid dynamics has certainly come of age in industrial applications and academic research. In the beginning, this popular field of study, usually referred to by its acronym CFD, was only known in the high-technology engineering areas of aeronautics and astronautics, but now it is becoming a rapidly adopted methodology for solving complex problems in modem engineering practice. CFD, which is derived from the disciplines of fluid mechanics and heat transfer, is also finding its way into important uncharted areas, especially in process, chemical, civil, and environmental engineering. Construction of new and improved system designs and optimization carried out on existing equipment through computational simulations are resulting in enhanced efficiency and lower operating costs. With the concerns about global warming and the world's increasing population, engineers in power-generation industries are heavily relying on CFD to reduce development and retrofitting costs. These computational studies are currently being performed to address pertinent issues relating to technologies for clean and renewable power as well as for meeting strict regulatory challenges, such as emissions control and substantial reduction of environmental pollutants. The fundamental basis of almost all CFD problems are the Navier–Stokes equations, which define any single-phase (gas or liquid, but not both) fluid flow. These equations can be simplified by removing terms describing viscous actions to yield the Euler equations. Further simplification, by removing terms describing vorticity yields the full potential equations. Finally, for small perturbations in subsonic and supersonic flows (not transonic or hypersonic) these equations can be linearized to yield the linearized potential equations.

3.1 Computational fluid dynamics can be summarized as follows

- An approximate numerical tool used to solve 3-D fluid flow problems whose solution has greater degree of convergence to the exact solution.
- An analyzer in optimizing the design procedure by constructing its own contingent objective function which is governed by some typical fluid flow parameters.
- A bundle of database helpful in simulating the flow characteristics of any fluid described in it.
- An authentic tool which avoids the tedious, long-winded calculations in determining the exact solution of various partial differential equations for a given boundary condition stipulated by the user.

3.2 Application of CFD

- In, designing Aircrafts and its aerofoiled shape wings, tail etc for given Mach number and the type of flow associated with it which may be sonic, sub-sonic, super-sonic.
- For, designing pumps and turbines by simulating and imparting the desired fluid flow conditions and characteristics.

- In Estimation and optimization of a design procedure pertinent to various fluid flow devices and machines.
- For designing and analyzing of the finned structure encapsulating the motor cycle engine.
- In the appraisal and analysis of complex structures by integrating with other simulation analysis programs aided by the ANSYS SOFTWARE.

3.3 CFD approach and problem solving methodology

There are generally three phases involved in the problem solving strategy adopted by any analysis problem. Each phase of this methodology is a rudimentary and an unavoidable procedure for having the desired outcome of best simulation results predicted by the software before-hand without an inclination to the destructive testing analysis. The following are the three phases associated with any ANSYS problem:

- Pre-Processing Phase:

- The geometry (physical bounds) of the problem is defined.
- The volume occupied by the fluid is divided into discrete cells (the mesh). The mesh may be uniform or non-uniform.
- The physical modeling is defined- for example, the equations of motion + enthalpy + radiation + species conservation.
- Boundary conditions are devised. This involves specifying the fluid behavior and its properties at the boundaries of the problem. For transient problems, the initial conditions are also defined.

- Processing Phase:

- The simulation is started and the equations are solved iteratively as a steady-state or transient. This phase is totally dependent upon the processing speed and type of processor employed in the work station.

- Post-Processing Phase:

- Finally a postprocessor is used for the analysis and visualization of the resulting solution.
- This phase usually constitutes various plots and graphs contingent to the fluid flow problem being analyzed.

The solver's analysis comes under the post processing phase of the CAD/CAM procedure after completing the Geometric modeling and physics definition of the problem. Actually, CFD is a misnomer for the computing solver in analyzing various fluid flow problems. But the actual tool or solver used for computing fluid flow problems is the ANSYS SOFTWARE and its embedded tools. ANSYS is an integrated software gleamed from various sources, database for performing wide range analysis, optimization of design which not only include fluid flow analysis but also convoluted problems associated with Civil structural analysis, Heat flow analysis, Dynamic analysis, Kinematic analysis, Electromagnetic problems etc. Various solvers incorporated in ANSYS for fluid analysis are:

- Fluid Flow- Blow Molding (Polyflow)
- Fluid Flow- Extrusion (Polyflow)
- Fluid Flow (CFX)
- Fluid Flow (Polyflow)

IV. Pre-processing phase:

Pre-Processing phase as described before deals with Geometric modeling, Meshing- mesh generation, mesh gradation, density, and lastly comes in defining the physics of the given fluid flow problem. The pre-processing phase serves as back bone of CAD/CAM processes and helps in providing better visual interface to the user integrating the graphics system along with the various periphery devices present in the work station. A typical workstation defining geometric modeling and mesh generation constitutes the following items:

- Graphics system and software.
- Periphery devices like-plotters, mouse, keyboard, monitor, digitizer, printers etc.
- CATIA software.

The pre-processing stage constitutes the crux of all the three phases describing the 3-D geometric model of the design process which helps in simulating the physics in the analysis stage. In addition it also helps in gridding the complex problem to a more facile one constituting the replica of the same element for quick generation of results. The pre-processing stage or phase comprises the following three indispensible elements:

- Geometric modeling
- Mesh generation
- Physics definition

The final facade of the drawing encompassing present day constructional features of a centrifugal pump is shown in the figure 4.1. The model includes an inlet, an outlet or discharge, impeller and a casing circumventing it and is made keeping in mind the present day constructional features of a centrifugal pump.

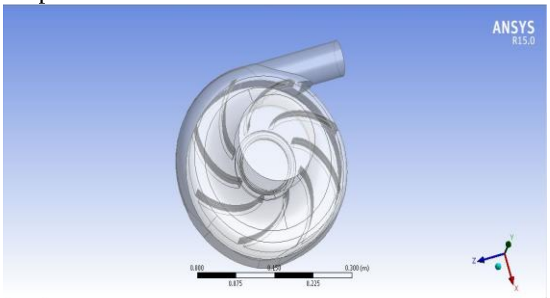

Fig 4.1

V. Processing phase:

Usually, the finite element approach comes under processing phase of the design procedure where the actual physics and boundary conditions are devised and analysis is carried giving adequate results. However, present day ANSYS software's simulating analysis program are capable of handling all the three phases of the design procedure namely-preprocessing, processing and post processing where intricate 3-D models are drawn and sketched with greater flexibility. Adding, meshing and grid generation is also incorporated in the software.

5.1 Steps in finite-element procedures

Most generalized procedure for carrying out FEA for any analysis problem is as follows:

- Procedure 1:*Discretization* Divide the object of analysis into a finite number of finite elements.
- Procedure 2:*Selection of the interpolation function* Select the element type or the interpolation function which approximates displacements and strains in each finite element.
- Procedure 3:*Derivation of element stiffness matrices* Determine the element stiffness matrix which relates forces and displacements in each element.
- Procedure 4:*Assembly of stiffness matrices into the global stiffness matrix* Assemble the element stiffness matrices into the global stiffness matrix which relates forces and displacements in the whole elastic body to be analyzed.
- Procedure 5:*Rearrangement of the global stiffness matrix* Substitute prescribed applied forces (mechanical boundary conditions) and displacements (geometrical boundary conditions) into the global stiffness matrix, and rearrange the matrix by collecting unknown variables for forces and displacements, say in the left-hand side, and known values of the forces and displacements in the right-hand side in order to set up simultaneous equations.
- Procedure 6:*Derivation of unknown forces and displacements* Solve the simultaneous equations set up in Procedure 5 above to solve the unknown variables for forces and displacements. The solutions for unknown forces are reaction forces and those for unknown displacements are deformations of the elastic body of interest for given geometrical and mechanical boundary conditions, respectively.
- Procedure 7:*Computation of strains and stresses* Compute the strains and stresses from the displacements obtained in Procedure 6

VI. Post-processing phase:

The solution, results and plots obtained in the ANSYS software has great significance for the pre-determined path chosen for analysis. Validation of the results can be done by Rapid prototyping before actual advent of the designed product into the market. In case of a civil structural problem, element strains and stresses are the final outcomes of the analysis along with typical plots depicting the stress distribution along the whole continuum. The results and plots obtained in the post-processing phase depends upon the following factors:

- Number of nodes, elements
- Type of element chosen
- Mesh density and mesh gradation
- Number of iterations

6.1 Significance of the plots

6.1.1 Absolute pressure rotation plot (impeller with casing)

The following adjacent plot relates the absolute pressure distribution along the domain of the centrifugal pump. The red portion depicts the area of high pressure prone region along the volute casing of the centrifugal pump and the navy blue portion entails the region of low absolute pressure.

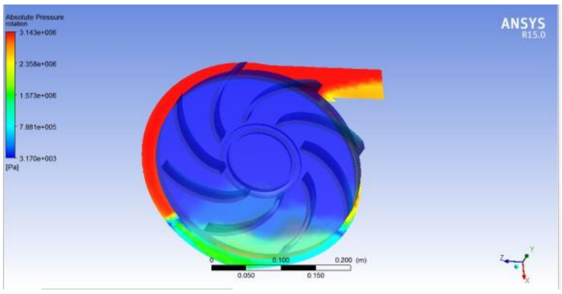

Fig 6.1.1 a

Obviously, it is true that the eye and the inlet of the impeller i.e the suction region has low pressure which is to be pressurized to a high pressure zone at the discharge at outlet. The absolute pressure chart maps the regions of the lowest pressure plots inorder to insinuate the presence of saturated vapor pressure at the eye of the impeller. This will serve as an indication of the cavitation in a pump. The areas of light blue also represents the scarcity of NPSH available at that point.

6.1.2 Pressure rotation plot (impeller pad only)

This lucidly depicts how pressure builds up along the plane of the impeller during the running conditions of the pump. The light blue portion encompassing some part of the eye is more prone to low absolute and have high risks of cavitation in those areas.

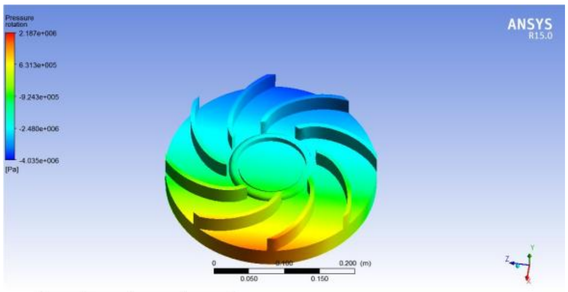

Fig 6.1.2 a

The impeller is given a specific angular speed and the pressure distribution plane can be observed clearly. It must be noted that cavitation tendancy depends greatly upon the speed of the impeller also. During rotation the outer edge of the impeller vanes are subjected to hydraullic stresses and muct be backed up with intricate contours (which is nothing but the casing) inorder to sustain it.

.6.1.3 Total pressure plot (impeller back portion and eye)

This plot gives the vulnerable areas of the high pressure at the interspaces enclosed between the vanes. The interspaces between vanes close to the discharge have high tendancy to hydraullic stresses (red region) during the running condition of the pump where as the interspaces included by the vanes at gradually increasing cross section of the casing is more prone to cavitation if the absolute pressure of the liquid is below vapor pressure of the liquid at ambient conditions. To have a better comprehension of the total pressure plot a perspective snapshot is

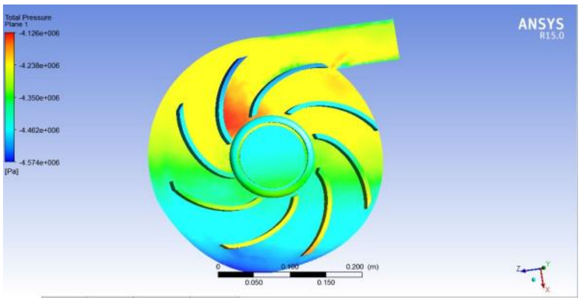

Fig 6.1.3a

taken which is schematically depicted in the folowing adjacent Fig 6.1.3 b. The hazy red portion

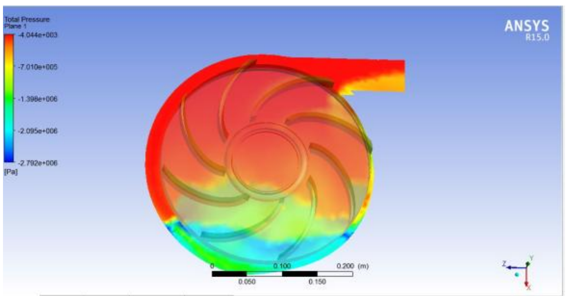

Fig 6.1.3 b

represents the casing encompassing the impeller and the discharge tube at outlet. The superimposition of the impeller plane and casing plane gives the above plot and can be validated as more generic form of interpretation for judging the centrifugal pump's performance.

6.1.4 Total pressure outlet

This plot enables the manufacturers to curtail and evaluate the performance plots of the pump depending upon the type of application demanded by the consumer. The difference between the inlet and outlet absolute pressure head determines the net

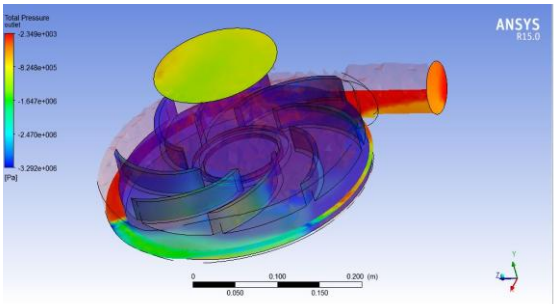

Fig 6.1.4 a

effective head developed by the pump at various speeds and operating suction head. The superimposition of all the above plots gives the following plot shown in figure 6.1 e which is used by the consumer in the manufacturer's catalogue for choosing a particular type of pump for a given application. The manufacturer's catalogue constitutes a scale entailing NPSH for a given speed and discharge head enabling the customer's to have a greater overview of the pump's commercial exploitation.

6.1.5 Vapor superfical velocity

The most exquisite feature of CFX solver of the ANSYS software is its ability to map and point the areas of highest vapor velocity due to the cavitation phenomenon causing pitting action on the casing and vane material. The vapor velocity flow pattern for a given speed of the impeller can be easily interpreted and comprehended. Even though the fluid attains particular pressure energy at the expense velocity head, there will always be certain disturbances in the flow due to the advent of cavitation leading to the fall of pressure energy. This case is most important pertinent to water which is making its way out of the impeller with cavitating conditions. Consequently, high velocity vapor bubbles start hitting the casing encompassing the impeller, ensuing pitting action. The adjacent plot gives the vapor superfical velocity at cavitating conditions and infers the places likely to effect greatly by pitting action of the high velocity vapor bubbles. The velocity of superficial vapor bubbles greatly depends on the type of casing and the angular velocity of the impeller. The red- dashed lines showcase the regions with high vapor velocity for a given speed of the impeller.

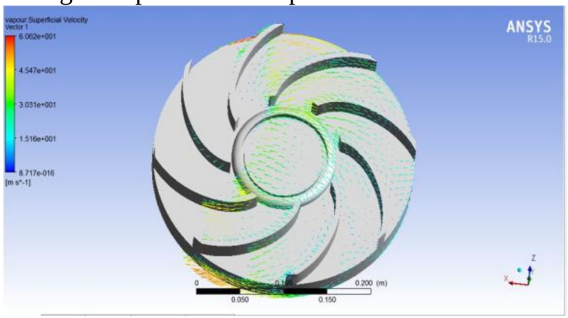

Fig 6.1.5 a

VII. Conclusion and scope:

There are a number of inferences made from the plots articulated by the CFD analysis of a centrifugal pump. Conclusive suppositions are brought down by contrasting the post-processing plots with different operating conditions of the centrifugal pump. The focus should be on resolving cavitation problems either by increasing the external pressure on the fluid or decreasing its vapor pressure. The external pressure could be increased by:

- Increasing the pressure at the pump suction such that the absolute pressure of water at inlet does not falls below the saturation pressure at that temperature present in the ambience.
- Reducing the energy losses incurring at the eye of the pump. This can be resolved by using a frictionless small length suction inlet.
- Using a larger pump.

Vapor pressure of the fluid is decreased by:

- Lowering the temperature of the fluid.
- Changing to a fluid with a lower vapor pressure.

We are witnessing a renaissance of computer simulation technology in many industrial applications. This changing landscape is partly attributable to the rapid evolution of CFD techniques and models. For example, state-of-the-art models for simulating complex fluid-mechanics problems, such as jet flames, buoyant fires, and multi-phase and/or multi-component flows, are now being progressively applied, especially through the availability of multi-purpose commercial CFD computer programs. The increasing use of these programs in industries makes clear that very demanding practical problems are now being analyzed by CFD. With decreasing hardware costs and rapid computing times, engineers are increasingly relying on the reliable yet easy-to-use CFD tools to deliver accurate results, as described in the examples in the previous sections. Additionally, significant advances in virtual technology and electronic reporting are allowing engineers to swiftly view and interrogate the CFD predictions and to make necessary assessments and judgments on a given engineering design. In industry, CFD will eventually be so entrenched in the design process that new product development will evolve toward “zero-prototype engineering.'' Such a conceptual design approach is not a mere flight of the imagination, but rather a reality in the foreseeable future, especially in the automotive industry. Looking ahead, full-vehicle CFD models with undershoot climate control, and external aerodynamics will eventually be assembled into one comprehensive model to solve and analyze vehicle designs in hours instead of days. Time-dependent simulations will be routinely performed to investigate every possible design aspect. Other related “co-simulation" areas in ascertaining the structural integrity as well as the acoustics of the vehicle will also be computed concurrently with the CFD models. Engineering judgment will be consistently exercised on the spot through *real-time* assessments of proposed customized design simulations in selecting the optimum vehicle. In the area of research, the advances in computational resources are establishing large eddy simulation (LES) as the preferred methodology for many turbulence investigations of fundamental fluid-dynamics problems. Since all real- world flows are inherently unsteady, LES provides the means of obtaining such solutions and is gradually replacing traditional two-equation models in academic research. The demand for LES modeling is steadily growing. LES has made significant inroads, especially in single-phase fluid flows. In combustion research, LES has also gained much respectability, particularly in capturing complex flame characteristics because of its better accommodation of the unsteadiness of the large-scale turbulence structure affecting the combustion

VIII. Limitations:

Although much effort has been focused on developing more robust CFD models to predict complicated multi-phase physics involving gas-liquid, gas-solid, liquid-solid, or gas-liquid-solid flows, LES (large eddy simulation) remains in its infancy of application to these flow problems. Instead, two-equation turbulence models are still very prevalent in accounting for the turbulence within such flows. LES may be adopted as the preferred turbulence model for multi-phase flows in the future but, in the meantime, the immediate need is to further develop more sophisticated two-equation turbulence models to resolve these flows. Based on current computational resources, numerical calculations performed through LES can be long and arduous due to the large number of grid nodal points required for computations. However, the ever-escalating trend of fast computing will permit such calculations to be performed more regularly in the foreseeable future. Also, with the model gradually moving away from the confines of academic research into the industry environment, it is not entirely surprising that LES will eventually become a common method for investigating many physical aspects of practical industrial flows.

8.1 Following are the limitations of CFD analysis in this project

- The rate of convergence to the exact solution of the partial differential equations is very low below 100 iterations and high speed computers are required to enhance and interpret iterations above 100. Generally, best solution arrives at 400-500 iterations and greatly depends upon the type of problem chosen.
- The remedies suggested for cavitation of the centrifugal pump relents on the operational working conditions of the pump and no constructional optimization details is disclosed by the project thesis.
- Cavitation analysis made in this project is confined to both on-load and off-load phases. However, it is difficult to infer conclusions from the on-load characteristics because it is contingent upon various factors like type of the casing material, vane material, thermal properties of the vanes and casing adhering to frictional head loss resulting in highly capricious NPSHr (net positive suction head required).
- No methods or remedies are studied regarding the erroneous vibrations and noise emanated from the working of a centrifugal pump.

IX. Nomenclature:

CFD- computational fluid dynamics.
NPSH- net positive suction head.
NPSHa- net postive suction head available.
NPSHr- net positive suction head rquired.
p1- absolute pressure at inlet.
pv1- vapor absolute pressure.
C- velocity at inlet.
p- density of liquid.
g- acceleration due to gravity.

X. Acknowledgment:

This project thesis and documentation is done with grim determination under strict tutelage of our mentor Dr. G. Rambabu and any claims raised by the third party for a facsimile unauthenticated copy is merely a coincidence and will be disavowed by the members of the batch. In addition, any attempts for emulating this thesis content and analysis will also be condemned and pertinent legal action will be sued against the third party. Finally we acknowledge the contribution of all those who have helped us directly or indirectly with their good wishes and constructive criticism which lead to successful completion of our Project.

References:

Journals:

[1] Reddy, J.N. (2005). *An Introduction to the Finite Element Method* (Third ed.). McGraw-Hill. journal.
[2] Hrennikoff, Alexander (1941). "Solution of problems of elasticity by the framework method". *Journal of applied mechanics* 8.4: 169–175.
[3] Elman; Howle, V; Shadid, J; Shuttleworth, R; Tuminaro, R et al. (January 2008). "A taxonomy and comparison of parallel block multi-level preconditioners for the incompressible Navier–Stokes equations". *Journal of Computational Physics* 227 (3): 1790–1808.
[4] Courant, R. (1943). "Variational methods for the solution of problems of equilibrium and vibrations". *Bulletin of the American Mathematical Society* 49: 1–23.
[5] Hirt, C.W.; Nichols, B.D. (1981). "Volume of fluid (VOF) method for the dynamics of free boundaries". Journal of Computational Physics.

Books:

[6] *CAD/CAM applications* by P.N.Rao.
[7] Shepard, Dennis G. (1956). *Principles of Turbomachinery*. McMillan.
[8] Reti, Ladislao; Di Giorgio Martini, Francesco (Summer 1963). "Francesco di Giorgio (Armani) Martini's Treatise on Engineering and Its Plagiarists".
[8] *Fluid mechanics and hydraulic* by Modi and Seth
[9] *Centrifugal pump design* by John Tuzson
[10] Hirt, C.W.; Nichols, B.D. (1981). "Volume of fluid (VOF) method for the dynamics of free boundaries". Journal of Computational Physics.
[11] Unverdi, S. O; Tryggvason, G.(1992). "A Front-Tracking Method for Viscous, Incompressible, Multi-Fluid Flows". J. Comput. Phys.
[12] Benzi, Golub, Liesen (2005). "Numerical solution of saddle-point problems". *Acta Numerica* 14:1–137.

.