Diyala Journal of Engineering Sciences

ISSN 1999-8716
Printed in Iraq

First Engineering Scientific Conference
College of Engineering –University of Diyala
22-23 December. 2010, pp. 170-180

CAVITATION IN CENTRIFUGAL PUMPS

Mohammed Khudhair Abbas
Engineering College , Diyala University

ABSTRACT:- Design, operation and refurbishment of centrifugal pumps are strongly related to cavitation flow phenomena, which may occur in either the rotating runner-impeller or the stationary parts of the centrifugal pumps. The numerical simulation (ANSYS CFX software Release 12) can be use to detect the cavitation in centrifugal pump. The total pressure at inflow boundary is reduced in small increment to meet the beginning cavitation.

The CFD results computed by ANSYS CFX software can be shown that the formation bubble form in a lower pressure area which caused by high velocity fluid. Inception cavitation occurs on the blade surface where the leading edge meets the tip. For lower NPSH values the cavitation zones move from leading to trailing edge. The drop in the head-NPSH curve begins when the cavity length is reached the maximum chord length of the blade. The computational grid was generated by SFX-Turbogrid with H-Grid through the blade and flow passage.

***Keywords*:-** Centrifugal pump, CFD, ANSYS CFX software Release 12, Cavitation

---

Nomenclature

D: impeller diameter (m)
g: gravitational acceleration (m/s^2)
H: head rise (m)
NPSH: net positive suction head (m)
NS: specific speed
P: static pressure (pa)
P_m : minimum pressure(Pa)
Q: flow rate (m^3/min)
u: is inlet tip speed (m/s)
r : radius (m)
Z : number of blade
ρ: density of water (kg/m^3)

Subscripts

$_{1:}$ impeller inlet
$_{2:}$ impeller exit
$_v$: vapor state

INTRODUCTION

Cavitation is an abnormal condition that can result in loss of production, equipment damage and worst of all, the term 'cavitation' comes from the Latin word cavus, which means a hollow space or a cavity. Webster's Dictionary defines the word 'cavitation' as the rapid formation and collapse of cavities in a flowing liquid in regions of very low pressure. In any discussion on centrifugal pumps various terms like vapor pockets, gas pockets, holes, bubbles, etc. are used in place of the term cavities. These are one and the same thing and need not be confused. The term bubble shall be used hereafter in the discussion. In the context of centrifugal pumps, the term cavitation implies a dynamic process of formation of bubbles inside the liquid, their growth and subsequent collapse as the liquid flows through the pump.

Motohiko (1) studied the cavitation flow in a low specific centrifugal pump speed by using two types of cavitation CFD codes. Okamura (2) used numerical cavitation prediction method available in commercial computational fluid dynamic software packages on the basis of a comparison of measurements obtained for a centrifugal pump. Nishi (3) used mini turbo-pump having its impeller diameter between 5 mm and 50 mm to find cavitation performance.

In this study flow field with cavitation in a centrifugal pump was computed for the validation of cavitation CFD. Several variable fringe plots on blade to blade view of inception cavitation are ploted to analyzed generation of vapor bubble .Location of maximum formation cavitation rate on the blade surface while NPSH reduced can be presented.

TYPE OF CAVITATIONS

In the context of centrifugal pumps, the term cavitation implies a dynamic process of formation of bubbles inside the liquid, their growth and subsequent collapse as the liquid flows through the pump. Generally, the bubbles that form inside the liquid are of two types: Vapor bubbles or Gas bubbles.

1. Vapor bubbles are formed due to the vaporization of a process liquid that is being pumped. The cavitation condition induced by formation and collapse of vapor bubbles is commonly referred to as Vaporous Cavitation as in figure (1). (4)
2. Gas bubbles are formed due to the presence of dissolved gases in the liquid that is being pumped (generally air but may be any gas in the system). The cavitation condition induced by the formation and collapse of gas bubbles is commonly referred to as Gaseous Cavitation. Both types of bubbles are formed at a point inside the pump where the local static pressure is less than the vapor pressure of the liquid (vaporous

cavitation) or saturation pressure of the gas (gaseous cavitation). Vaporous cavitation is the most common form of cavitation found in process plants. Generally it occurs due to insufficiency of the available NPSH or internal recirculation phenomenon. It generally manifests itself in the form of reduced pump performance, excessive noise and vibrations and wear of pump parts. The extent of the cavitation damage can range from a relatively minor amount of pitting after years of service to catastrophic failure in a relatively short period of time.

Gaseous cavitation occurs when any gas (most commonly air) enters a centrifugal pump along with liquid. A centrifugal pump can handle air in the range of 0.5% by volume. If the amount of air is increased to 6%, the pump starts cavitating. The cavitation condition is also referred to as air binding. It seldom causes damage to the impeller or casing. The main effect of gaseous cavitation is loss of capacity. (4)

There are three steps to occur the cavitation as in figure (2).

Step One, Formation of bubbles inside the liquid being pumped.

Step Two, Growth of bubbles.

Step Three, Collapse of bubbles.

CAVITATION MODEL

Cavitation is the process of the formation of vapor bubbles in low pressure regions within a flow. One might imagine that vapor bubbles are formed when the pressure in the liquid reaches the vapor pressure p_v of the liquid at the operating temperature. The static pressure in any flow is normally nondimensionl as a pressure coefficient, C_p defined as (5)

$$C_p = \frac{(p - p_1)}{0.5\rho u^2} \text{……………………} (1)$$

Where P_1 is inlet static pressure and u is inlet tip speed.

In the absence of cavitation, the fluid velocities and the pressure coefficient are independent of the level of the pressure, a change in the inlet pressure p_1, will simply result in an equal change in all the other pressures, so that C_p is unaffected. It follows that in any flow with prescribed fluid velocities, geometry and Reynolds number, there will be a particular location at which the pressure (p) is a minimum and that the difference between this minimum pressure $p_{\min}$ and the inlet pressure p_1 is given by

$$C_{p\min} = \frac{(p_{\min} - p_1)}{0.5\rho u^2} \ldots\ldots\ldots\ldots\ldots\ldots\ldots (2)$$

Where $C_{p\min}$ is some negative number which is a function only of the geometry of the centrifugal pump and the Reynolds number. It could be establish the value of the inlet pressure p_1, at which cavitation would first appear (assuming that this occurs when $p_{\min} = p_v$) as p_1 is decreased, namely

$$(p_1)_{\substack{cavitation\\ appearance}} = p_v + 0.5\rho u^2(-C_{p\min}) \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots (3)$$

It can be find cavitation number as

$$\sigma = \frac{(p_1 - p_v)}{0.5\rho u^2} \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots (4)$$

And net positive section head as

$$NPSH = \frac{(p_t - p_v)}{\rho g} + \frac{v^2}{2g} \quad \ldots\ldots\ldots\ldots\ldots (5)$$

And specific speed as

$$NS = \frac{\Omega Q^{0.5}}{(NPSH.g)^{0.75}} \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots (6)$$

Another nondimensional parameter, called Thoma's cavitation factor as

$$\sigma_{th} = \frac{(p_t - p_v)}{(p_2 - p_t)} \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots (7)$$

NUMERICAL IMPLEMENTATION

This study was carried out with a Computational Fluid dynamic (CFD) code ANSYS CFX software. The three dimensional geometry of centrifugal flow pump impeller consisted of blade profile, hub and shroud. H-Grid is used through the blade and the flow passage. The geometrical parameters for impeller are shown in table (1). Computational grid consist total number of nodes in the target domain 25368. The flow simulation is executed in the rotation frame of reference. Standard (k-ε) model is used for the turbulence. Turbulent wall treatment; the volume of fluid model is selected for cavitation model. Upwind difference is used for solution scheme.

COMPUTATION DOMAIN AND BOUNDARY CONDITIONS

The boundary condition consists of two walls, inlet and outlet at blade and hub surface, smooth relative frame stationary wall is selected as a default boundary condition. At shroud surface, relative frame counter rotating wall is applied with Log-Law wall model and smooth wall. Inlet boundary condition absolute total pressure is set include of specification of turbulent intensity and length scale. Direction of absolute flow is defined in Cartesian coordinate. At the outlet, a mass flow for one passage is specified.

The solver is run in steady state condition while upwind difference is selected for discretization scheme. Initial guesses of the flow field are estimated from one-dimensional analysis. Static pressure and velocity flow field are specified as initial guess.

To compute head drop curve, the absolute total pressure at inflow boundary is reduced from 100000 Pa to meet the onset of cavitation. After cavitations occur the simulation is run until the inlet total pressure is reduced to 30000 Pa.

RESULTS AND DISCUSSIONS

It can be use numerical simulation (ANSYS CFX software) to detect the cavitation in centrifugal pump. Figure (3) shows head drop curve as well as curve that computed by ANSYS CFX at design flow and rate speed. Volume of fluid model is selected as a cavitation model to compute rate of vapor bubble production. At inlet boundary total pressure is decreased to meet a status that cavitation formation starting.

Figure (4) shows cavity length with vapor fringe plot blade to blade view which corresponding to points A, B, C and E in head drop curve figure (3). There is no cavitation when NPSH = 8.78 m, attach cavitation when NPSH=7.795m and super cavitation when NPSH =2.07m as shown in figure (4). Therefore head drop curve in figure (3) can be divided according to cavity length on blade surface into three portions as no cavitation, attach cavitation and super cavitation. No cavitation is range from A to B in figure (3); in this portion rate of vapor production approximate zero. Attach cavitation is range from B to C that has cavity length on blade surface less than or equal chord length. Cavitation on suction surface is growing when decreasing NPSH. If cavity length grows up more than chord length, super cavitation occurs as shown in figure (4).Super cavitation is range from C to E in figure (3).

Figure(5) shows that the stages of water- vapor fringe plot meridional view at 99% span that the cavitation is form ,figure(5.1) no cavitation appear then develop to reach maximum at figure (5.4) when static pressure reduce below vapor pressure.(6)

To clarify cavitation phenomenon in flow passage in centrifugal pump, the compute static pressure is plotted in blade to blade view at NPSH value of (8.78 m) that inception cavitation occurred at the blade leading edge near the tip, figure (6) shows that the pressure fringe plot blade to blade view at 99% span in this region the static pressure reduce below vapor pressure when the flow field static pressure decrease below liquid vapor pressure, cavitation will be formation.

CONCLUSION

The flow through the blade passage was numerically studied with CFD code, ANSYS CFX in order to detect formation of cavitation in centrifugal pump. Head drop curve has a knee shape that head remain constant while NPSH decreased and head will be rapidly decreased at critical point at point D. The beginning of cavitation in the blade passage can be detected and shown in quality and quantity with numerical simulation.

The inception of cavitation is take place on the suction surface where the leading edges meet the tip. In pressure distribution plot shows that the cavitation zone expanding to the trailing edge especially in super cavitation case.

The available NPSH of the system must be equal to or greater than the NPSH required by the centrifugal pump in order to avoid cavitation difficulties.

REFERENCES

1. Motohiko Nohmi 2003" Cavitation CFD in A Centrifugal Pump" Fifth International Symposium on Cavitation Osaka, Japan.
2. Tomoyoshi Okamura 2003" Cavitating Flow Calculations in Industry" International Journal of Rotating Machinery, 9(3): pp 163–170.
3. Michihiro Nishi 2003"Cavitation Performance of a Centrifugal Impeller Suitable for a Mini Turbo-Pump" Fifth International Symposium on Cavitation Osaka, Japan.

4. Mukesh Sahdev 2008 " Centrifugal Pumps: Basic Concepts of Operation, Maintenance, and Troubleshooting" associate content writer sahdevm@iocl.co.in.
5. Christopher E. Brennen, 1994," Hydrodynamics of Pumps" California Institute of Technology, USA.
6. Miguel Asuaje 2005 " Numerical Modelization of the Flow in Centrifugal Pump: Volute Influence in Velocity and Pressure Fields." International Journal of Rotating Machinery.

Table (1): The geometrical parameters for impeller

Impeller dimension	
Inlet diameter at hub D1h (mm)	36
Inlet diameter at tip D1t (mm)	80
Outlet width b2 (mm)	4.4
Blade inlet angle at hub β1h (°)	21.5
Blade inlet angle at tip β1t (°)	31.1
Blade outlet angle β2 (°)	22.5
Blade number z	6

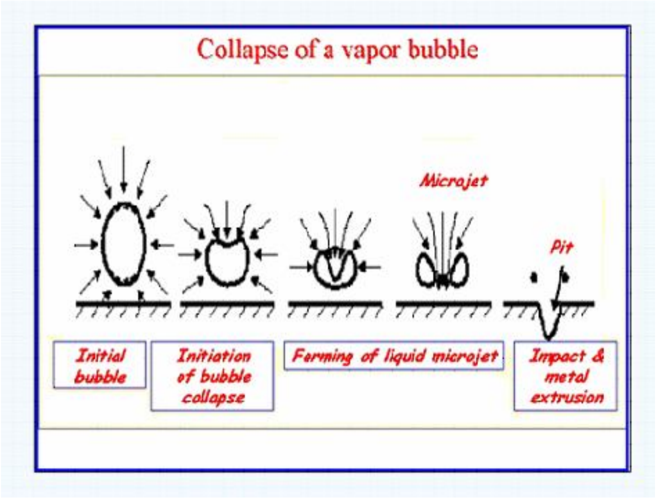


Fig. (1): Collapse of a vapor bubble [4].

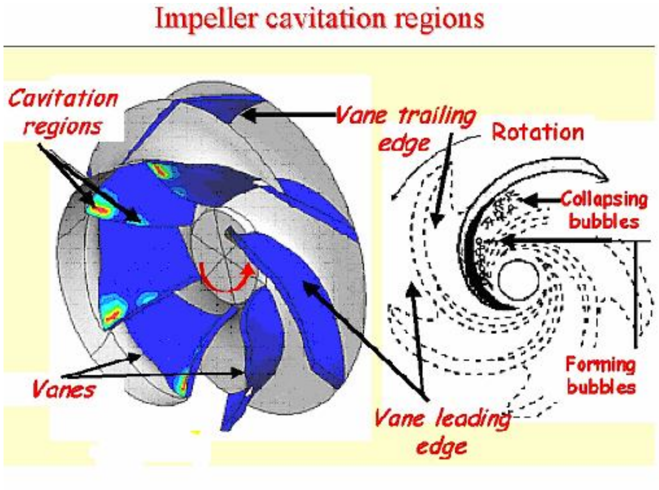


Fig. (2): Impeller cavitations regions [(4)].

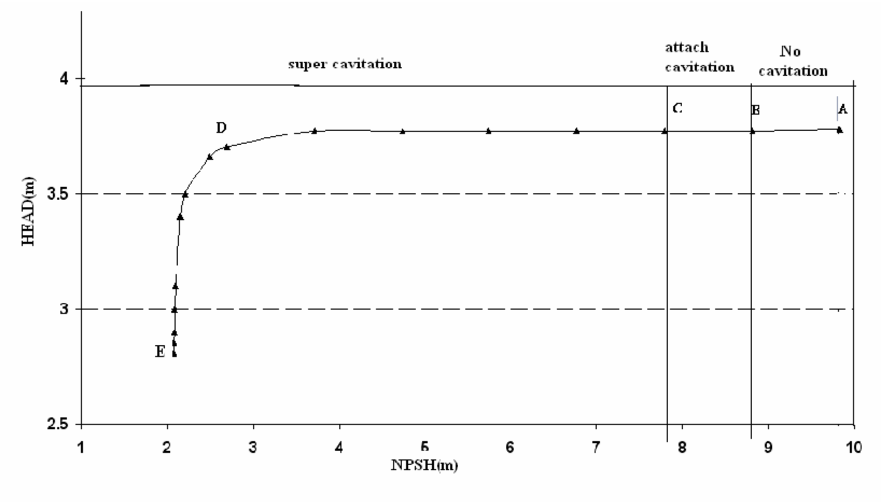
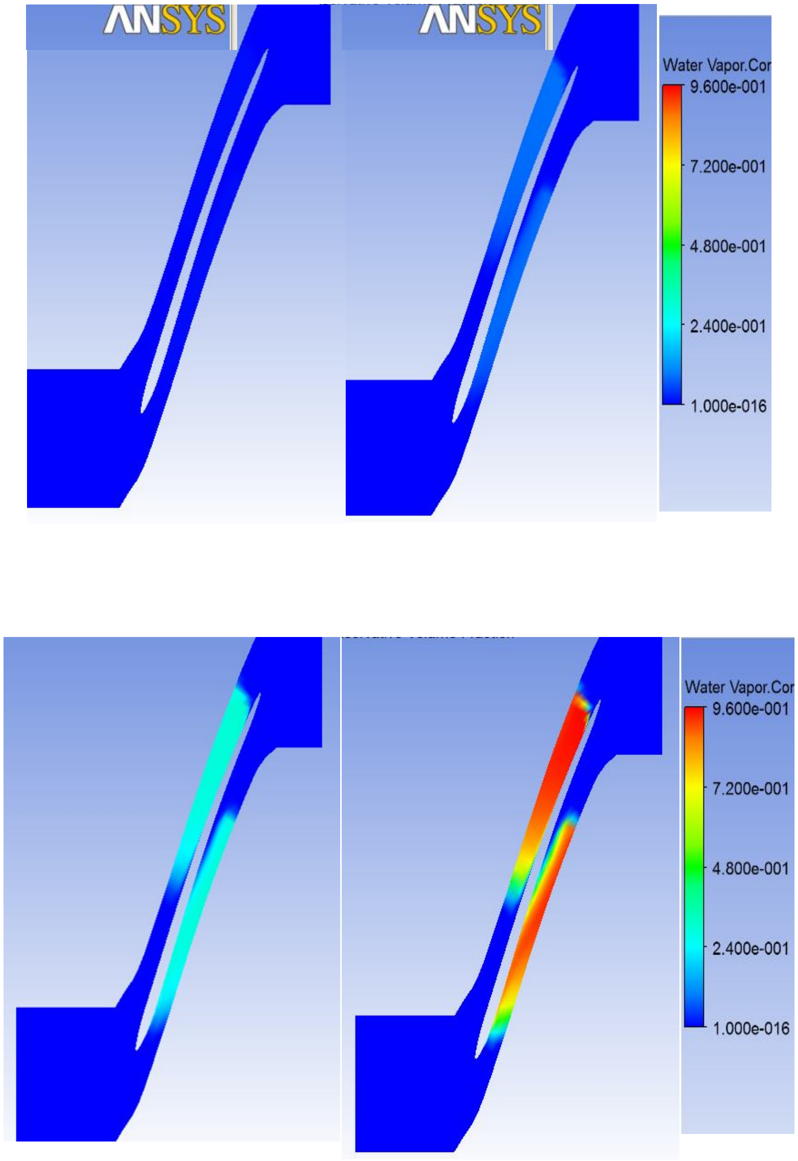


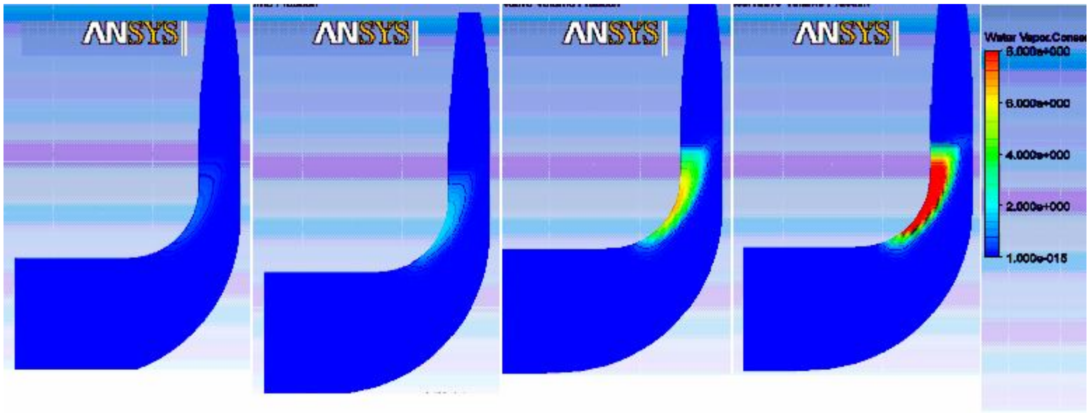
Fig. (3): NPSH-Head curve at flow rate 16 Kg/s and speed 3000 rpm**.**



NPSH=9 NPSH= 8.78

NPSH=7.795 NPSH=2.07

Fig. (4): Water- vapor fringe plot blade to blade view at 99% span**.**



5.1 **5.2** **5.3** **5.4**

Fig. (5): Water- vapor fringe plot meridional view at 99% span.

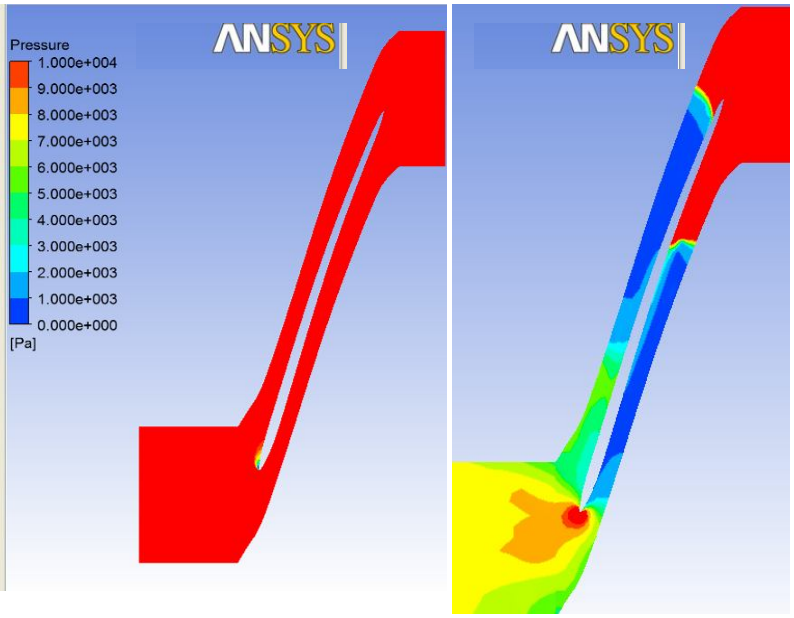


Fig. (6): Pressure distribution fringe plot blade to blade view at 99% span at Head Drop.

التكهف في المضخات الانتباذية

محمد خضير عباس
مدرس مساعد
كلية الهندسة . جامعة ديالى

الخلاصة

ان تصميم وعمل وصيانة المضخات الانتباذية (الطاردة من المركز) مرتبط بشدة بظاهرة جريان الفجوات (التكهف) والتي تحصل في الاجزاء المتحركة اوالثابته للمضخات. تم اسخدام محاكاة عددية باستخدام برنامج الـ ANSYS لاكتشاف التكهف في المضخات الانتباذية. تم تقليل الضغط الكلي المتدفق بزيادات صغيرة لحين ظهور الفقاعات. النتائج المستحصلة CFD اظهرت ان تكوين الفقاعات يتشكل في المناطق ذات الضغط القليل بسبب سرعة المائع العالية . في البداية ، التكهف يحصل في نهاية الحافة الامامية لسطح الريشة ولقيم قليلة من NPSH مناطق التكهف تنتقل من الحافة الامامية الى الحافة الخلفية لسطح الريشة، الهبوط في مخطط الضغط - NPSH يبدأ عندما يصل طول الفقاعة يصل الى طول اكبر وتر للريشة الخاصة بالمضخةالانتباذية .